NÖHÜ Müh. Bilim. Derg. / NOHU J. Eng. Sci., 2025; 14(1), 289-296

Niğde Ömer Halisdemir Üniversitesi Mühendislik Bilimleri Dergisi
Niğde Ömer Halisdemir University Journal of Engineering Sciences

Araştırma makalesi / Research article

www.dergipark.org.tr/tr/pub/ngumuh / www.dergipark.org.tr/en/pub/ngumuh

Computational and experimental investigation of water surface around bridge piers

Köprü ayakları etrafındaki su yüzeyinin sayısal ve deneysel olarak incelenmesi

Mesut Çelik[1,*], Mücella İlkentapar[2*], Serhat Akşit[3] Ahmet Alper Öner[4]

[1,2,4] *Erciyes University, Civil Engineering Department, 380039, Kayseri, Türkiye*
[3] *Abdullah Gül University, Civill Engineering Department, 380039, Kayseri, Türkiye*

Abstract

Free surface flow around bridge piers is one of the most important problems of open channel hydraulics. The structural design of the bridge should be done correctly along with the hydraulic design by simulating the water surface around the bridge piers together with the scour that will occur around the bridge piers. If the hydraulic design of the bridge piers is not good enough, it can cause very serious damages in situations such as floods and overflows. The aim of this paper is to analyze the effect of bridge piers on the water surface experimentally and numerically. For the numerical analysis, FLOW-3D, a computational fluid dynamics package program that performs 3-dimensional solution, and HEC-RAS package program are used and the results are compared. In the study, flows with the same velocity for different number of bridge openings were studied and compared with the experimental results. In addition, the FLOW 3D package program, which performs solutions with the finite volume method, was compared with the experimental results by performing solutions with different mesh sizes. As a result, it was observed that the solutions performed with the FLOW-3D package program showed very close results with the experimental data.

Keywords: Open channel hydraulic, Bridge piers, Water surface, FLOW-3D

Özet

Köprü ayakları etrafındaki serbest yüzeyli akım, açık kanal hidroliğinin en önemli problemlerinden biridir. Köprünün yapısal tasarımı, köprü ayakları etrafında oluşacak oyulma ve köprü ayakları etrafındaki su hareketini simüle eden hidrolik tasarım ile birlikte doğru bir şekilde yapılmalıdır. Köprü ayaklarının hidrolik tasarımı yeterince iyi değilse, sel ve taşkın gibi durumlarda çok ciddi hasarlara neden olabilir. Bu çalışmanın amacı köprü ayaklarının su yüzeyine etkisini deneysel ve sayısal olarak analiz etmektir. Sayısal analiz için hesaplamalı akışkanlar dinamiği ile 3 boyutlu çözüm yapan FLOW-3D paket programı ile HEC-RAS paket programı kullanılmış ve sonuçlar karşılaştırılmıştır. Çalışmada farklı köprü açıklık sayıları için aynı hızda akışlar incelenmiş ve deneysel sonuçlarla karşılaştırılmıştır. Ayrıca sonlu hacimler yöntemi ile çözüm yapan FLOW 3D paket programında farklı mesh boyutları ile çözüm yapılarak deneysel sonuçlar ile karşılaştırılmıştır. Sonuç olarak FLOW-3D paket programı ile gerçekleştirilen çözümlerin deneysel veriler ile oldukça yakın sonuçlar gösterdiği gözlemlenmiştir.

Anahtar Kelimeler: Açık kanal hidroliği, Köprü ayakları, Su yüzü profilleri, FLOW-3D

1 Introduction

With the progress of civilization, the development of transportation networks has come to a very important point from past to the present and has become a very important part of human life, especially after the industrial revolution. One of the most important cornerstones of these transportation networks is the bridges built over rivers and open channels. Over time, the need to cross wider spans led to the necessity of placing bridge piers in the middle of rivers or open channels. Bridge piers can cause significant changes in water flow, leading to floods. Floods, which we consider as natural disasters, can have devastating effects in our country and around the world During the flood, the water in the river overflows from its own bed, damaging the surrounding farmlands, roads, living points, etc. Flood can cause significant damage to people's social life, and it can also cause direct death and injury. Accurate determination of the water surface profiles during flooding is extremely important in terms of minimizing the damages that may ocur.

Bridges should have a good hydraulic design together with a good structural design. Hydraulic engineers have to consider the problems posed by the structures located on open channels. Therefore, a good bridge hydraulic design should include the analysis of the relationship between water surface profiles and flow rates in order to foresee the effects of structure on water. For example, the piers of a bridge whose structural design has been made with a high safety factor but the hydraulic design has been disregarded will narrow the flow cross-sectional area. The hydraulic changes caused by this narrowing can cause scours around bridge piers and after a while these scours can cause a substantial destruction.

In cases which there is no water structure on the river, methods such as Direct Step Method [1] and Standard Step

* Sorumlu yazar / Corresponding author, e-posta / e-mail: mucella@erciyes.edu.tr. (M. İlkentapar)
Geliş / Recieved: 23.08.2024 Kabul / Accepted: 23.12.2024 Yayımlanma / Published: 15.10.2025
doi: 10.28948/ngmuh.1537850

Method [2] can be used to determine the water surface profile. However, in cases which there is a structure on the river, the determination of the water surface profile becomes more complicated. The flow around bridge piers has been investigated experimentally and theoretically by many researchers in the past [3-12].Two types of changes are observed in the open channels and rivers where the bridge structures are located. The first type of change is the local scours that occur at the bridge piers. These scours may reduce the stabilization of bridges and cause collapses in the future. Scour around bridge piers have been studied by Laursen[13] and various works. The second type is the afflux which is the main subject of this study. Afflux is the rise of water because the minimum specific energy level in the narrowing section is greater than the specific energy level of the flow. The level of afflux can be affected by parameters as flow rate, the geometry of the bridge, the span of the bridge piers and the geometry of the piers, etc. The water surface profiles that we investigate these affluxes which are the most important data used in flood control studies that will be caused by structures or bridges on the rivers. Many methods are used to obtain these data. The most used of them are; D'Aubuisson, Weisbach, Nagler, Lane, Yarnell, Kindswater, Izzard, Biery, and Delleur, Bradley (USBPR) methods [14]. In addition to these, Energy and Momentum methods are used.

When a bridge is built on an open channel, it narrows the cross section of the channel and obstruct the flow so that energy loss occurs. To overcome this energy loss, the upstream water level rises to a higher level than normal. This additional elevation is called afflux. After the water passes the bridge, the water level first decreases, and after a while it returns to its first condition (Figure 1)

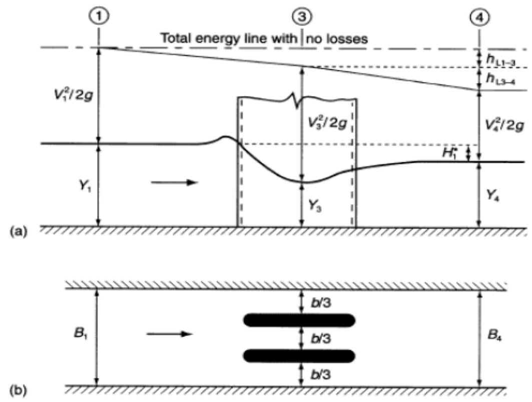

Figure 1. (a) Schematic representation of the rise of the water surface profile at the bridge entrance with a longitudinal section of uniform flow at normal depth, (b) Plan view of the flow (Les Hamill, Bridge Hydraulics) [15])

The effect of bridge piers on the flow has been investigated for more than 150 years. One of the earliest studies on this is d'Aubuisson's study in 1840 [3]. Later, Nagler (1918) [4] and Yarnell (1934) [5]also conducted research on bridge piers. These studies were generally of similar type. These types were experimental studies carried out in smooth flow condition using long laboratory channels. One or more of the same type of bridge piers are positioned in the central part of the channel so that their effects on the channel can be measured. As a result of these studies, they came up with equations for finding water heights.

In this study, the numerical analysis results are validated by using the past experimental studies. These experimental results are presented in the study of "The Investıgatıon Of Backwater Caused By Bridges On The River Experimental And Wıth HEC-RAS" [16], which was previously carried out by Arzu Hadi at Erciyes University Hydraulics Laboratory in January 2017. Besides, the analysis results of the HEC-RAS program from the same study is used for comparison with FLOW-3D analysis results. These experiments were performed in an open channel made of glass measuring 0.6x0.6x9.5 m in Erciyes University Engineering Faculty Hydraulic Laboratory.. Experiments were carried out for 6 different flow rates in 4 different bridge spans (M=b/B= 0.83, 0.75, 0.67 and 0.58). Here, B represents the channel width, while b denotes the bridge span. The water surface profiles resulting from these experiments are modeled for a single flow rate (21.23 l/sec) under the same conditions in the FLOW-3D program and compared with the water surface profile analysis data.

With the development of technology and the advancement of programming languages, many package programs are used for providing analysiss with the help of these methods. Programs such as HEC-RAS, ISIS, and PHS are examples. In this study, the change of water surface profiles for three different opening conditions at a single flow rate will be calculated numerically using the Flow 3D package software. FLOW-3D commercially; It is a Computational Fluid Dynamics software used for different models and different purposes. Computational Fluid Dynamics; It is a technique of simulating the entire operation of the stream by providing analysis for each discretized cell separately using the Continuity Equation and the Navier-Stokes Equations. Using Computational Fluid Dynamics software is similar to experimental work in many ways. If the experimental work is modeled incorrectly with respect to reality, the results do not represent the real situation. Similarly, numerical modeling cannot show the real situation when entered incorrectly. (FLOW-3D user manual) [15] FLOW-3D software, developed and sold by Flow Science Incorporation which founded by C.W. Hirt in 1980, was released in 1985. FLOW-3D software, which performs the numerical analysis of the water surface based on the Volume Of Fluid [15] approach, which was added to the literature by C. W. Hirt, performs numerical analysis on Computational Fluid Dynamics. Recently, many researchers have published hydraulic analyzes with the FLOW-3D [17-25], but there are few studies on bridge hydraulics with the FLOW-3D software[26,27].

2 Methodology

2.1 Experimental study

Experiments in Arzu Hadi's work [27] were carried out in Erciyes University Engineering Faculty Civil Engineering Department Hydraulics Laboratory in a 60x60 cm and 9.5 m long channel. The construction material of the channel is glass. In the study, 4 different bridge types were used for 4 different flow conditions. All bridges have square feet and the features that distinguish bridges from each other are the number of spans. It was built with 4 different spans as one span, two spans, three spans and four spans, and experiments were carried out on these bridges. Factors such as the stabilization of the flow regime, the formation of the free surface profile, the examination of vortex shedding and separation zones, and the development of the boundary layer play a significant role in determining the placement of bridge piers. Considering all these factors, the bridges were placed 5.5 meters from the beginning of the 9-meter channel, and the effect of this bridge on the water surface profile was investigated. The data obtained in these experiments were compared with the data calculated by modeling in the HEC-RAS package program with 4 different methods (Energy, Momentum, Yarnell, WSPRO). In this study, the flow field will be numerically modeled with the FLOW 3D package program for one-two and three-opening bridge cases as shown in Figure 2. And 21.23 lt/sec value was used as the flow rate. The obtained data will be compared with the experimental studies of Arzu Hadi for the same flow conditions and the HEC-RAS package program data.

2.2 Bridge hydraulic modelling

In the modelling section of FLOW-3D, CGS (Centimeter-Gram-Second) unit system was used and free surface and uncompressed flow were entered as flow properties. The analysis time varied according to the models and was continued until it became stable. Since the channel used in the experiment has a slope of 0.001, the slope must be entered in the modeling, but the slope cannot be entered for the existing floor in the Flow-3D program. The slope value can only be given to the movement of the water. That situation can only be achieved by the vectorial distribution of the gravitational acceleration according to the slope. For this reason, in the gravity section, the values of 0.981 for the X component, 0 for the Y component, and -980.99 for the Z component are entered and inclination is given in the plane. The k-ε turbulence model is preferred in numerical simulations due to its simplicity, computational efficiency, robustness, and consistency with experimental results. It is suitable for modeling flows around cylinders, bridge piers, and other hydraulic structures where flow separation, vortex shedding, and turbulent boundary layers are critical phenomena. The k-ε model remains one of the most reliable and efficient tools for engineering applications where computational cost, time, and accuracy are all critical considerations. For all these reasons, the k-ε turbulence model was selected as the turbulence model.

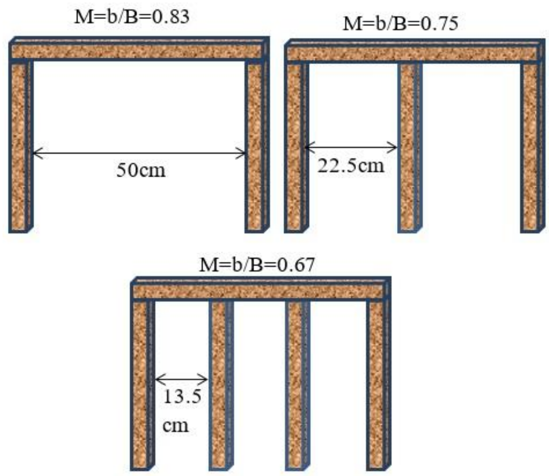

Figure 2. Bridge types in the experimental study to be used in this study

When starting the modeling process, first of all, the analysis region was created. Mesh stops were made according to the bridge geometry and a general analysis region was created. Since the bridges were positioned at the 5.50 m in the experiments, a second mesh block was also created, since the mesh density would be made between the 5 meters and 6 meters in the horizontal direction from the beginning, and a constriction zone was made in that block. After the bridge was created, all models were divided into cells and the mesh structure was created. While determining the mesh structure, computations were made in 5 different mesh structures on one span bridge to find the mesh variation with the lowest error rate, and the mesh structure with the lowest error rate was used for other bridge situations. In the study, it was observed that the mesh structure with the lowest error rate was the constricted mesh structure, the information of which will be given below. Mesh spacing was determined as 0.50 cm in the x direction for the whole computation zone, and 1 cm in the y and z directions, and 0.50 cm in all ratios in the constriction zone. In total, analysiss were made with the help of 3660000 meshes.

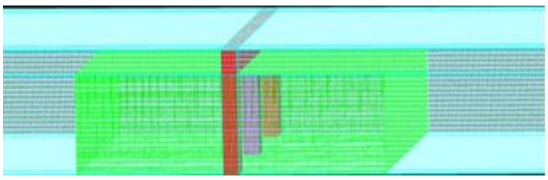

Figure 3. 3D view of constricted mesh zone of 3-pier bridge

3 Results and discussion

Arzu Hadi, who made the experimental study based on this study, in her study, saw that the computations made with the Energy, Momentum and Yarnell methods gave more accurate results than the WSPRO Method, and as a result, she published the analysis results made with the Energy Method mostly in the study, since there was no big difference between the other methods for comparative graphs [2]. In

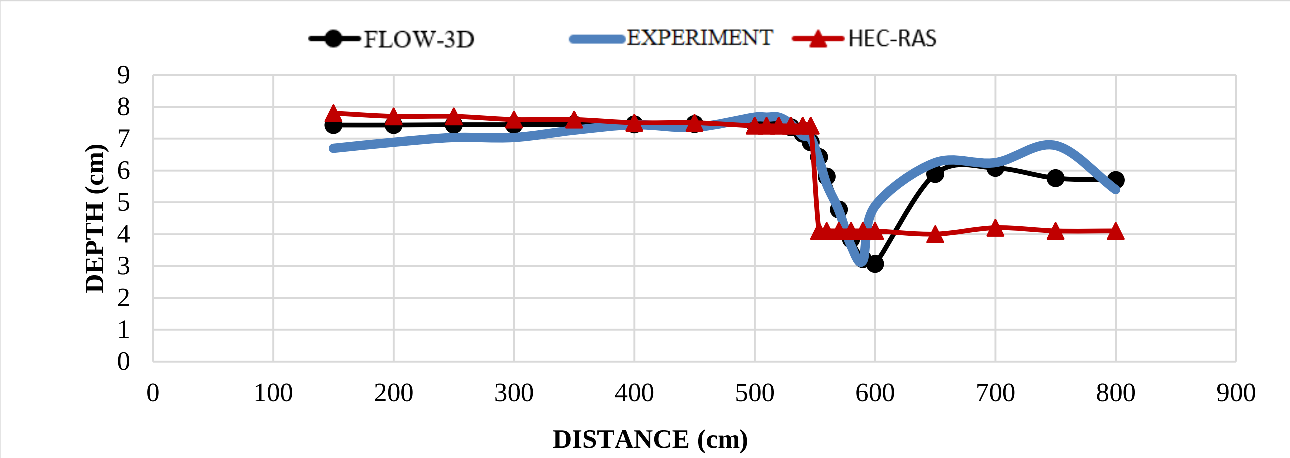

Figure 4. Water surface profiles for single span bridge

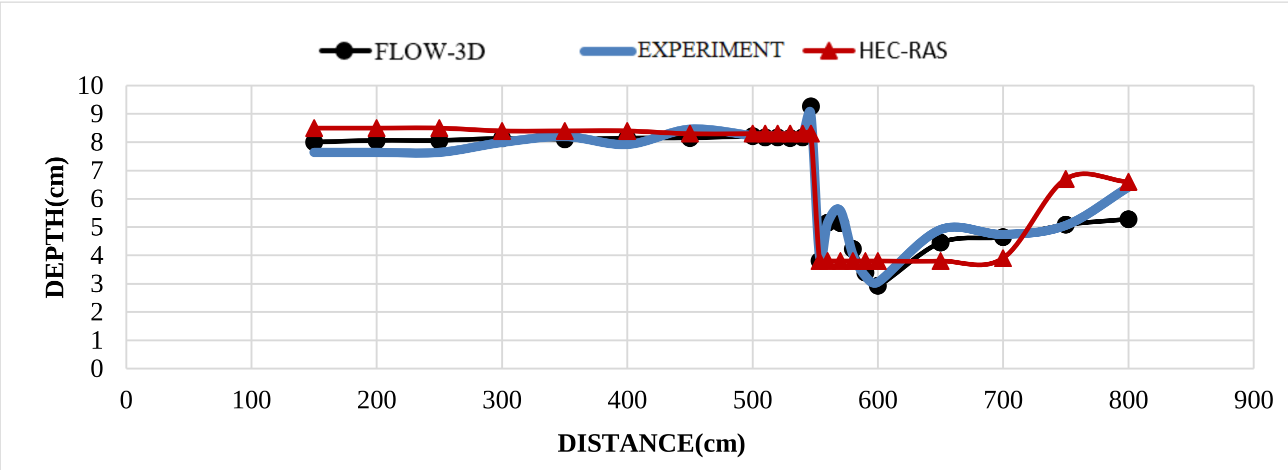

Figure 5. Water surface profiles for two span bridge

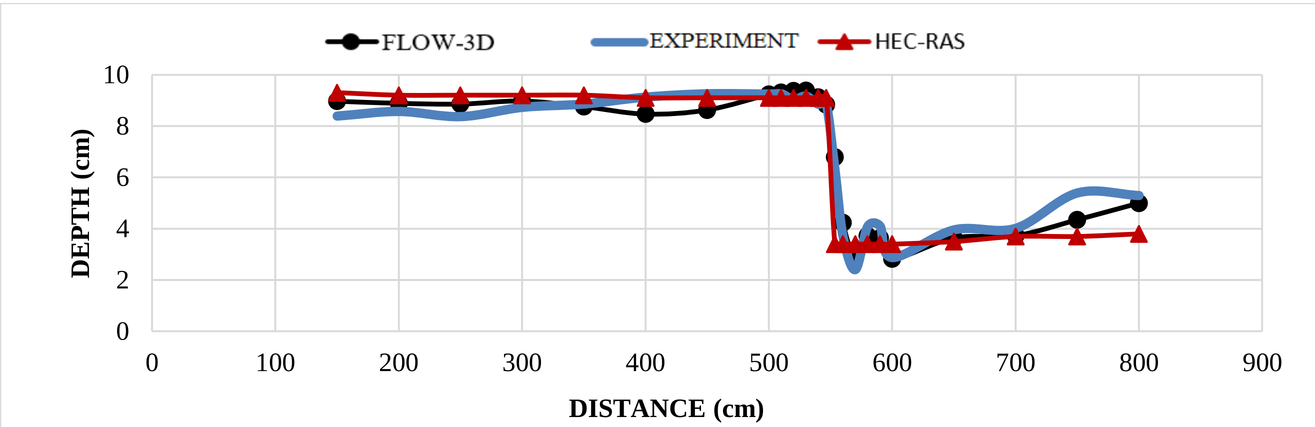

Figure 6. Water surface profiles for three span bridge

this study, when comparing the data of the HEC-RAS package program, the results of the computation made with the Energy Method will be used. . The flow through the bridge piers was calculated by dividing meshes with the Finite Volume Method, using the twoequation (k-ε) turbulence model. Calculated values were compared with experimental measurements and HEC-RAS analysis results for 3 different bridge cases. Values taken from 23 different sections are given in the table. Comparisons were made by subtracting separate differences for each section on the table. In the comparison, the average of the differences of all sections was also calculated. While calculating the error rates, the formula as $\%Error = \left|\frac{WS_o - WS_c}{WS_o}\right| * 100$ was applied. Here, WS_o shows the observed experiment result and WS_c shows the computation result. Comparisons were made with both tables and graphs of water surface profiles.. While making the graphic drawings, the drawings were not made according to the real scales, and the depth section was enlarged and the graphs were drawn in order to display it more clearly.

When Table 1 is examined, the average error rate in the calculations made with the HEC-RAS program is 14.77%, while the average error rate in the calculations made with the

Table 1. Comperative numerical results for single span bridge

Distance X (m)	Flow Depths Measured (cm)	Hec-Ras (Energy Method) (cm)	FLOW-3D	Difference (%) FLOW-3D	Hec-Ras
0.000	6.41				
1.000	6.88				
1.500	6.70	7.8	7.43	10.90	16.42
2.000	6.89	7.7	7.43	7.84	11.76
2.500	7.04	7.7	7.44	5.68	9.37
3.000	7.04	7.6	7.44	5.68	7.95
3.500	7.27	7.6	7.45	2.48	4.54
4.000	7.44	7.5	7.45	0.13	0.81
4.500	7.36	7.5	7.46	1.36	1.90
5.000	7.68	7.4	7.48	2.60	3.65
5.100	7.68	7.4	7.47	2.73	3.65
5.200	7.68	7.4	7.43	3.26	3.65
5.300	7.47	7.4	7.35	1.61	0.94
5.400	7.10	7.4	7.15	0.70	4.23
5.465	7.03	7.4	6.88	2.13	5.26
5.535	6.36	4.1	6.42	0.94	35.53
5.600	5.56	4.1	5.81	4.50	26.26
5.700	4.75	4.1	4.77	0.42	13.68
5.800	3.65	4.1	3.84	5.21	12.33
5.900	3.17	4.1	3.21	1.26	29.34
6.000	4.88	4.1	3.06	37.30	15.98
6.500	6.25	4.0	5.89	5.76	36.00
7.000	6.25	4.2	6.08	2.72	32.80
7.500	6.79	4.1	5.76	15.17	39.62
8.000	5.40	4.1	5.70	5.56	24.07
AVR				**5.48**	**14.77**

In the FLOW-3D program, the average error rate between 5.1 and 5.9 meters, which is the region where the flow reaches stability, was calculated as 5.48%. Compaction zone, decreased to 2.28%. Considering the analysis made in the HEC-RAS program, the error rate varied before and after the bridge. In the HEC-RAS results, the average error rate on the upstream side is 5.70%, while the mean error rate on the downstream side is 26.56%. When the graph in Figure 4 is examined, it is seen that the results of the HEC-RAS package program are uniform on the upstream and downstream sides and do not show the upstream and downstream fluctiations in the test results. It is seen that the results of the FLOW-3D program, on the other hand, are quite close, although they do not exactly match the lines of the test results. It was observed that both numerical analysis gave values close to the experimental results in the afflux zone. According to the test results, while the maximum flow height in the affluxing region was 7.68 cm, it was determined as 7.48 cm in the FLOW-3D analysis and 7.40 cm in the HEC-RAS analysis

When Table 2 is examined, the average error rate in the analysis made with the HEC-RAS program is 10.71%, while the average error rate in the analysis made with the FLOW-3D program is 3.71%. In the FLOW-3D program, the average error rate decreased to 2.67% in the range of 5.10 and 5.90 meters, which is the compaction zone.

Table 2. Comperative numerical results for two span bridge

Distance X (m)	Flow Depths Measured (cm)	Hec-Ras (Energy Method) (cm)	FLOW-3D	Difference (%) FLOW-3D	Hec-Ras
0.000	7.00				
1.000	7.64				
1.500	7.64	8.5	8.00	4.71	11.26
2.000	7.64	8.5	8.06	5.50	11.26
2.500	7.64	8.5	8.06	5.50	11.26
3.000	7.99	8.4	8.13	1.75	5.13
3.500	8.20	8.4	8.1	1.22	2.44
4.000	7.92	8.4	8.14	2.78	6.06
4.500	8.46	8.3	8.15	3.66	1.89
5.000	8.23	8.3	8.21	0.24	0.85
5.100	8.23	8.3	8.17	0.73	0.85
5.200	8.23	8.3	8.17	0.73	0.85
5.300	8.23	8.3	8.15	0.97	0.85
5.400	8.23	8.3	8.17	0.73	0.85
5.465	9.00	8.3	9.26	2.89	7.78
5.535	3.97	3.8	3.81	4.03	4.28
5.600	5.16	3.8	5.15	0.19	26.36
5.700	5.59	3.8	5.14	8.05	32.02
5.800	4.04	3.8	4.23	4.70	5.94
5.900	3.27	3.8	3.39	3.67	16.21
6.000	3.05	3.8	2.93	3.93	24.59
6.500	4.91	3.8	4.45	9.37	22.61
7.000	4.74	3.9	4.65	1.90	17.72
7.500	5.06	6.7	5.08	0.40	32.41
8.000	6.41	6.6	5.28	17.63	2.96
AVR				**3.71**	**10.71**

Considering the analysis made in the HEC-RAS program, the error rate varied before and after the bridge. In the HEC-RAS results, the average error rate on the upstream side is 4.72%, while the mean error rate on the downstream side is 18.51%. When the graph in Figure 5 is examined, it is seen that the results of the HEC-RAS package program are uniform on the upstream and downstream sides, similar to the one-span bridge, and do not show the hydraulic jumps and fluctuations in the test results. It is seen that the results of the FLOW-3D program, on the other hand, are quite close, although they do not exactly match the lines of the test results, as in the single-span bridge. It was observed that both numerical analysiss gave values close to the experimental results in the afflux zone, but the values of FLOW-3D were closer to the experimental measurements. According to the test results, while the maximum flow height in the afflux zone was 9 cm, it was observed as 9.26 cm in the FLOW-3D analysis and 8.30 cm in the HEC-RAS analysis.

When Table 3 is examined, the average error rate in the analysiss made with the HEC-RAS program is 12.04%, while the average error rate in the analysis made with the FLOW-3D program is 6.54%. In the FLOW-3D program, the average error rate increased to 7.44% in the range of 5.10 and 5.90 meters, which is the compaction zone.

Table 3. Comperative numerical results for three span bridge

Distance	Flow Depths			Difference (%)	
X (m)	Measured (cm)	Hec-Ras (Energy Method) (cm)	FLOW -3D	FLOW -3D	Hec-Ras
0.000	7.00				
1.000	7.64				
1.500	8.39	9.3	8.97	6.91	10.85
2.000	8.57	9.2	8.89	3.73	7.35
2.500	8.37	9.2	8.86	5.85	9.92
3.000	8.73	9.2	8.98	2.86	5.38
3.500	8.86	9.2	8.76	1.13	3.84
4.000	9.13	9.1	8.47	7.23	0.33
4.500	9.26	9.1	8.63	6.80	1.73
5.000	9.25	9.1	9.24	0.11	1.62
5.100	9.26	9.1	9.31	0.54	1.73
5.200	9.03	9.1	9.38	3.88	0.78
5.300	9.17	9.1	9.39	2.40	0.76
5.400	8.85	9.1	9.13	3.16	2.82
5.465	8.85	9.1	8.84	0.11	2.82
5.535	6.31	3.4	6.79	7.61	46.12
5.600	3.71	3.4	4.24	14.29	8.36
5.700	2.40	3.4	2.93	22.08	41.67
5.800	4.08	3.4	3.72	8.82	16.67
5.900	4.10	3.4	3.63	11.46	17.07
6.000	2.87	3.4	2.82	1.74	18.47
6.500	3.95	3.5	3.64	7.85	11.39
7.000	4.01	3.7	3.73	6.98	7.73
7.500	5.39	3.7	4.35	19.29	31.35
8.000	5.29	3.8	5.00	5.48	28.17
AVR				**6.54**	**12.04**

Considering the analysiss made in both programs, the error rate varied highly before and after the bridge. In the HEC-RAS results, the average error rate on the upstream side is 3.84%, while the mean error rate on the downstream side is 22.70%. In the FLOW-3D results, the average error rate on the upstream side is 3.44%, while the mean error rate on the downstream side is 10.56%. When the graph in Figure 6 is examined, it is seen that the results of the HEC-RAS package program are similar to the first two bridge sections, and the results of the HEC-RAS package program are uniform on the upstream and downstream sides and do not show the hydraulic jumps and fluctuations in the test results. However, contrary to the first two sections, it was seen that the analysis of the HEC-RAS program gave closer values for the three-span bridge than the first two cases. As seen in the first two sections, the results of the FLOW-3D program appear to be quite close to the lines of the test results, although they do not exactly match. It was observed that both numerical analysiss gave values close to the experimental results in the afflux zone. According to the test results, while the maximum flow height in the afflux region was 9.26 cm, it was observed as 9.39 cm in the FLOW-3D analysis and 9.10 cm in the HEC-RAS analysis.

While using the FLOW-3D program, 2 types of mesh regions were used and the average error rates were calculated for these two separate mesh regions. In Table 4, these error rates are given for both mesh regions. Mesh 1 represents the general mesh zone, and mesh 2 represents the constricted mesh zone.

When Table 4 is examined, while the compaction region gives more accurate results for the first two case bridge sections, the compacted region gives more inaccurate results in the case of a three-span bridge. The reason for this is that there are bridge piers in the constriction region, and since the program has difficulty in analyzing the current passing through the three-span bridge, the constrction region has given inaccurate results.

Table 4. Average differences for mesh zones

Bridge Types	Mesh 1	Mesh 2
Single Span Bridge	7.94	2.28
Two Span Bridge	4.51	2.67
Three Span Bridge	5.84	7.44

On the other hand, in this study, different mesh patterns were tried to determine the correct mesh variation for the single-span bridge, and accordingly, the mesh pattern with the least error rate was used in the other stages of the study. Table 5 shows the average error rates, times of becoming steady and computation times of all tested mesh patterns for the single span bridge.

Table 5. Results for mesh patterns in different variations

Pattern types	Difference	Computation times	Times of becoming steady	Total number of meshes
Large mesh	5.84	1 h 29 min	22 sec	358160
Very large mesh	5.60	59 min	40 sec	227392
Wide constricted	6.62	52 h	44 sec	5580000
Standart mesh	5.78	4 h 2 min	52 sec	2700000
Constricted mesh	5.48	18 h	50 sec	3660000

When Table 5 is examined, it is seen that both the computation times are prolonged and there are no significant improvements in the difference rates if the mesh constriction is made too wide.

4 Conclusions

As a result; When all three bridge cases are compared, it has been observed that the FLOW-3D program analysiss for most parameters produce values closer to the experimental results than the HEC-RAS program analysiss. Both program analysiss gave the lowest error rates in the two-span bridge section. The FLOW-3D program gave the highest error rate in the case of a three-span bridge, while the HEC-RAS program gave the highest error rate in the case of a single-span bridge. These error rates are given in Table 6.

Table 6. Average Differences

Bridge Types	FLOW-3D	HEC-RAS
Single Span Bridge	5.48	14.77
Two Span Bridge	3.71	10.71
Three Span Bridge	6.54	12.04

It has been observed that the FLOW-3D program, which solves with the help of the Finite Volume Method using the twoequation (k-ε) turbulence model, gives more accurate results than the HEC-RAS program, which solves the water surface profiles using the Energy, Momentum, Yarnell and WSPRO methods in cases where there are bridge piers in open channel flows. For this reason, the FLOW-3D program is recommended for bridge pier problems in open channel flows. When solving with the FLOW 3D program, it is recommended that the mesh patterns should be made without too much constricting, taking into account the computation processes.

Conflict of interest

The authors declare that there is no conflict of interest.

Similarity Rate (Turnitin): %5

References

[1] M. H. Chaudry, Open Channel Flow. New York: Springer Science+Business Media, 2008.

[2] F. M. Henderson, Open Channel Flow. New York: MacMillan Company, 1966.

[3] J.F.D.A. Voissins, Traité d'hydraulique: à l'usage des ingénieurs. Pitois-Levrault, 1840.

[4] D. L. Yarnell, Bridge piers as channel obstructions. No. 442. US Department of Agriculture, 1934.

[5] P. Dubuat, Principes d'hydrauliques vérifiés par un grand nombre d'expériences,1786.

[6] P. F. Biery and J. W. Delleur, Hydraulics of single span arch bridge construction. Journal of the Hydraulics Division, 88.(2), 75-108, 1962. https://doi.org/10.1061/JYCEAJ.0000711

[7] G.V. Skogerboe, Comparison of Bridge Backwater Relations. Journal of the Hydraulics Division, 99.6, 921-938, 1973. https://doi.org/10.1061/JYCEAJ.0003665

[8] G. Seckin, R. Yurtal, and T. Haktanir, Contraction and expansion losses through bridge constrictions. Journal of Hydraulic Engineering 124.5,546-549, 1998. https://doi.org/10.1061/(ASCE)0733-9429(1998)124:5(546)

[9] F. Ahmed, and N. Rajaratnam, Flow around bridge piers. Journal of Hydraulic Engineering, 124.3,288-300, 1998. https://doi.org/10.1061/(ASCE)0733-9429(1998)124:3(288)

[10] J. Hunt, G.W. Brunner and B. E. Larock, Flow transitions in bridge backwater analysis. Journal of Hydraulic Engineering, 125.9, 981-983, 1999. https://doi.org/10.1061/(ASCE)0733-9429(1999)125:9(981)

[11] R.J. Charbeneau and E.R. Holley, Backwater effects of bridge piers in subcritical flow. USA: Center for Transportation Research, Bureau of Engineering Research, University of Texas at Austin, 2001.

[12] E.M. Laursen, Bridge design considering scour and risk. Transportation Engineering Journal of ASCE 96.2, 149-164, 1970. https://doi.org/10.1061/TPEJAN.0000079

[13] K.J. Kaatz and W. P. James Analysis of alternatives for computing backwater at bridges. Journal of Hydraulic Engineering, 123.9, 784-792, 1997. https://doi.org/10.1061/(ASCE)0733-9429(1997)123:9(784)

[14] C. W. Hirt and B. D. Nichols, Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of computational physics, 39.1 ,201-225, 1981. https://doi.org/10.1016/0021-9991(81)90145-5

[15] F.S. İnc., Flow-3D User's Manuals, 2017.

[16] A.M.W.M.H.M. Hadi, Akarsu Üzerindeki Köprülerin Neden Olduğu Kabarmanın Deneysel Ve Hec-Ras İle İncelenmesi, Erciyes Üniversitesi, Yüksek Lisans Tezi, 2017.

[17] L.G. Castillo and J. M. Carrillo, Comparison of methods to estimate the scour downstream of a ski jump. International Journal of Multiphase Flow, 92, 171-180, 2017. https://doi.org/10.1016/j.ijmultiphaseflow.2017.03.006

[18] N. Soydan, O. Şimşek, and M. Aköz, Prediction and validation of turbulent flow around a cylindrical weir. European water, 57, p. 85-92, 2017.

[19] J. Vives-Costa, J.J. De-Felipe, and E. Pena-Pitarch, Numerical model for a nineteenth-century hydrometric module. Journal of irrigation and drainage engineering, 145.11, 2019. https://doi.org/10.1061/(ASCE)IR.1943-4774.0001415

[20] E.H.H. Al-Qadami, et al, Numerical modelling of flow characteristics over sharp crested triangular hump. Results in Engineering, 4, 2019. https://doi.org/10.1016/j.rineng.2019.100052

[21] R. Kaurav and P.K. Mohapatra, Studying the peak discharge through a planar dam breach. Journal of Hydraulic Engineering, 145, 6, 2019. https://doi.org/10.1061/(ASCE)HY.1943-7900.0001613

[22] A. M. Helmi, H. T. Essawy, and A. Wagdy, Three-dimensional numerical study of stacked drop manholes. Journal of Irrigation and Drainage Engineering, 145, 9, 2019. https://doi.org/10.1061/(ASCE)IR.1943-4774.0001414
[23] M. İlkentapar and A. A. Öner, Geniş Başlıklı Savak Etrafındaki Akımın İncelenmesi. Niğde Ömer Halisdemir Üniversitesi Mühendislik Bilimleri Dergisi, 6.2, 615-626, 2017. https://doi.org/10.28948/ngumuh.341819
[24] H. K. Jalal and W. H. Hassan, Three-dimensional numerical simulation of local scour around circular bridge pier using Flow-3D software, IOP Conference Series: Materials Science and Engineering, Vol. 745, No. 1, IOP Publishing, 2020. https://doi.org/10.1088/1757-899X/745/1/012150
[25] M. Ghasemi and S. Soltani-Gerdefaramarzi, The scour bridge simulation around a cylindrical pier using Flow-3D, Journal of Hydrosciences and Environment, 1.2, 46-54, 2017. https://doi.org/10.22111/jhe.2017.3357
[26] L. Hamill, Bridge Hydraulics, CRC Press, 1998.
[27] A.M.W.M.H.M. Hadi, Akarsu Üzerindeki Köprülerin Neden Olduğu Kabarmanın Deneysel Ve Hec-Ras İle İncelenmesi, Erciyes Üniversitesi, Yüksek Lisans Tezi, 2017.