

## **Experimental and numerical modelling of bridge piers having different number of axis**

GOKNUR ELIF YARBASI KAYHANLAR, ALPASLAN YARAR\*

*Konya Technical University, Konya, Turkiye, gknrelfyarbasi@gmail.com*

*\*Corresponding author: ayarar@ktun.edu.tr, Turkiye*

*Received:*  
10.08.2023

*Accepted:*  
11.12.2023

*Published:*  
20.12.2023

### **Abstract**

*The interaction of bridge piers and current with each other is mutual. As a result of this interaction, it has been determined that one of the main causes of damage to bridges is the scouring of bridge piers. One of the factors on which the scouring procedure depends is the flow velocity. For this reason, in this study, the velocity distributions that occur before the bridge piers, which are positioned as single axis and double axis, were examined physically and numerically. In experimental studies, the reading values obtained by using ADV were compared with the results obtained from the ANSYS-Fluent package program used in numerical modeling. The performance of the model used and the effects of placing the piers as single axis and double axis were evaluated. As a result of the study, it was determined that the ANSYS used in determining the flow velocity area was very compatible with the experimental results and that the flow velocity acting in the system obtained by placing the bridge piers as a single axis are larger.*

**Keywords:** *bridge Piers, experimental modelling, numerical modelling*

### **INTRODUCTION**

Bridges are one of the hydraulic structures built on rivers and open channels. One of the most important issues in the design of these bridges is the effect of the bridge piers on the flow and the other is the effects of the flow on the bridge piers. As a result of these effects, it has been determined that one of the main causes of damage to bridges is the scouring of bridge piers. Scour is bed erosion caused by hydrodynamic forces. The rise from the flow velocity causes a series of scour density [1]. Scour is a concept that varies depending on many factors. We can say that the main reason for the scouring of a bridge pier is the vortices in various regions. In general, the scouring structure varies depending on the material thickness, Reynolds number, geometrical shape of the piers and flow velocity [2]. Because of scour, the stability of the bridge piers decreases and the piers collapse by becoming unable to carry the bridge. In order to solve this problem, it is necessary to determine the most appropriate pier type for the stream regime or to determine whether the stream regime can be adapted to the existing pier type. It is very important to determine the velocity distributions around the bridge pier in order to choose the appropriate pier for the river structure and make designs with minimal damage.

The investigation of the velocity changes around bridge piers dates back to the 1960s. Richardson and Panchang [3], examined the speed changes occurring around the bridge piers and the effects of these changes on the structure. Ashgriz et al. [4] numerically modeled the flow interacting with a half-cylindrical structure using the ANSYS package program. They obtained the numerical water surface profiles by using the Volume of Fluid (VOF) method and showed the pressure and velocity distributions obtained from the numerical solutions graphically. Barbhuiya and Dey [5] made measurements around a half-cylindrical foot adjacent to the channel wall in a rectangular channel.

In these studies, they determined the values of velocity, turbulence intensity, turbulence kinetic energy and stress as a function of time. The three-dimensional turbulent flow field around the bridge abutment was measured in the laboratory using an Acoustic Doppler Velocimetry (ADV). Gümüş et al. [6] physically and numerically investigated the flow interacting with the trapezoidal head structure, which creates a narrowing and widening section in the open channel. In the experimental studies, the velocity field was measured using an Acoustic Doppler Velocimeter (ADV) and the water surface profile was measured using a limnimeter. Flow velocity area and water surface profile were determined by using Standart  $k-\epsilon$  (SKE), Renormalization Group  $k-\epsilon$  (RNG), Realizable  $k-\epsilon$  (RKE), Standart  $k-\omega$  (SKW), Shear Stress Transport (SST), Reynolds Stress Model (RSM) models and Detached-Eddy Simulation (DES), Large Eddy Simulation (LES) in numerical models. As a result of the study, it was determined that the DES model was more successful than the other models used in determination of the flow velocity area and free water surface profile. Huang et al. [7] numerically analyzed the scale effect and scour in turbulent flow around the bridge abutment. Fayyadh et al. [8] modeled a diagonal bridge using the finite element method for different flow rates. Soydan et al. [9] measured the velocity distribution of the 3D turbulent flow around the bridge abutment and compared it with ANSYS-Fluent numerical results. Zhu and Liu [10] used computational fluid dynamics method in numerical simulations to identify and evaluate the local scour pit around the cylindrical bridge pier. Afzal et al. [11] used the three-dimensional computational fluid dynamics (CFD) method to calculate the scour and sedimentation around the bridge pier for two different boundary conditions, constant flow rate and regular waves. Some other types of hydraulic structures were also investigated both experimentally and numerically. Hager and Chwalt [12] experimentally examined the flow over a broad-crested weir using different flow rates. Chanson and Montes [13] conducted experimental studies on the flow passing through the circular weir. Yıldız and Yazar [14] examined the head over broad-crested weirs, having different slopes, experimentally and numerically. Fathi-Moghaddam et al. [15] conducted some studies on gabion weirs to investigate the hydraulic parameters of flow by using a numerical model. Also, they validate results of numerical model by using data get from laboratory experiments. Kocaer and Yazar [16] investigate the flow over Ogee spillway by performing experiments and simulating numerical models. Han et al. [17] determined the effectiveness of a Surface Flow Constructed Wetlands (SFCWs) in removing suspended solids based on the predicted flow characteristics and distribution of suspended solids in the wetland. Song and Zhou [18] developed a numerical model that may be applied to analyze the 3D flow pattern of the tunnel or chute spillways, particularly the inlet geometry effect on flow condition.

In this study, the velocity distributions around the bridge piers, which are positioned as single axis and double axis, were examined physically and numerically. In experimental studies, the reading values obtained by using ADV were compared with the results obtained from the ANSYS-Fluent package program used in numerical modeling. The performance of the model used and the effects of placing the piers as single axis and double axis were evaluated.

## **MATERIALS AND METHODS**

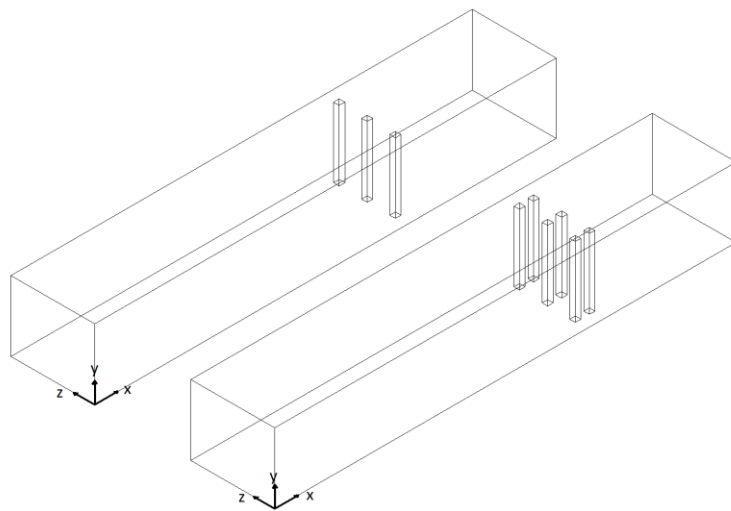
Physical model experiments were carried out in a 6.5 m long, 0.5 m deep and 0.3 m wide glasses-walled open channel in the Hydraulics laboratory of Konya Technical University. Fig. 1 shows the experimental setup in the laboratory and its schematic representation.

As can be seen in Fig. 2, it is placed the smooth square bridge piers of  $0.03 \times 0.03$  m, 2.30 m away from the beginning of the channel as single axis and double axis.

In the experimental studies, velocity distributions were measured by using an Acoustic Doppler Velocimeter (ADV). Thanks to the frequency changer control panel, different discharges were obtained were measured the velocity distributions at a distance of approximately 10 cm from the piers for four different discharge values.



**Fig. 1.** Open channel system used in experiments and schematic view



**Fig. 2.** Location of bridge piers

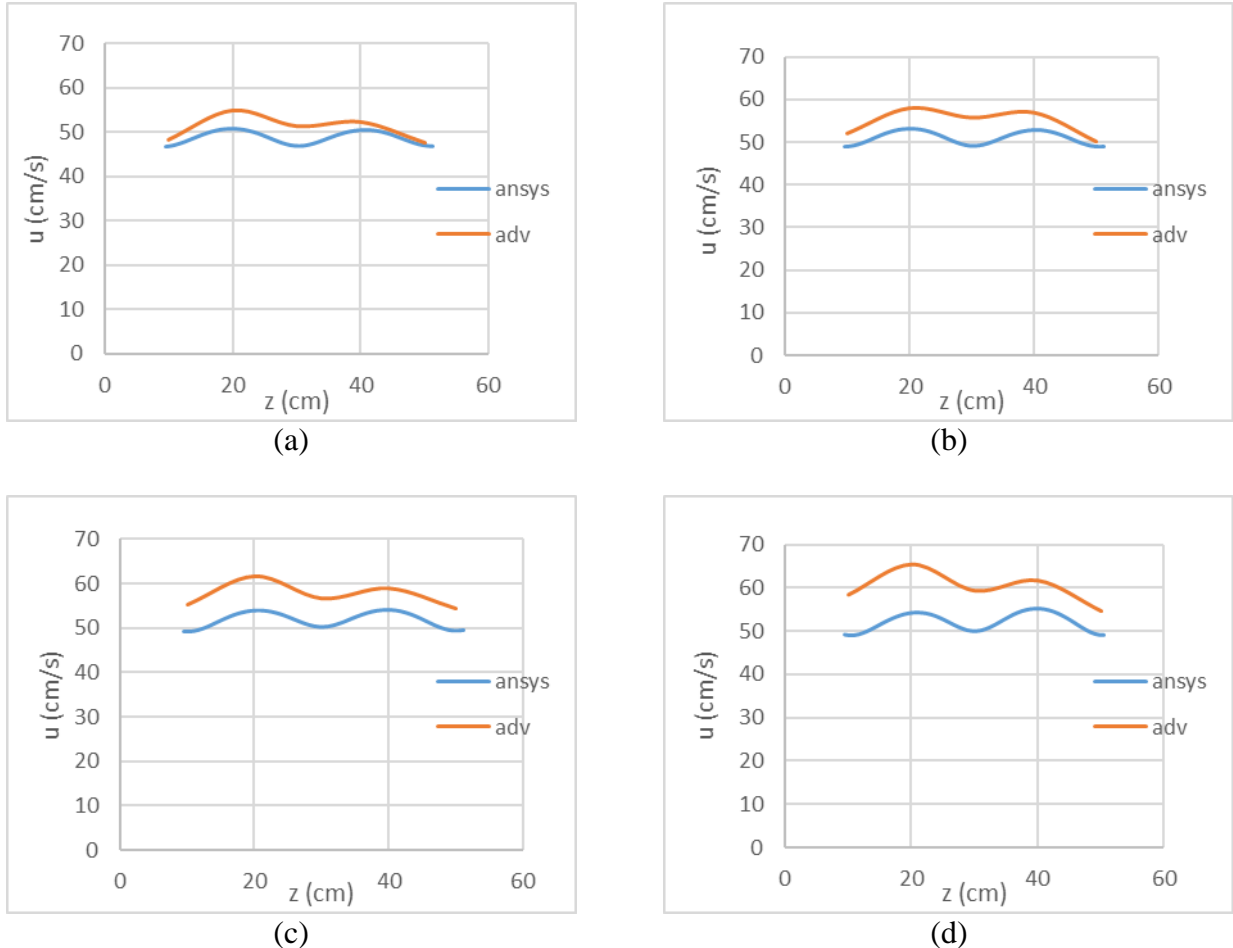
Due to the difficulties in reading the data precisely, only the velocity distributions in the x-direction could be used for comparison purposes. ANSYS-Fluent package program was used for numerical investigations. Fluent is a powerful tool for analyzing and optimizing the movement and interaction of fluids in real-world applications. CFD is based on Navier-Stokes equations, which is a mathematical modeling of fluid motion [19]. This program, which is used to simulate the flow, consists of several steps. The first one is the creation of the three-dimensional flow field by means of an auxiliary program and transfer to the ANSYS-Fluent. The second step is the creation of the mesh network of the model, which is the most important step for the precise solution.

The third step is to start the analysis after the boundary conditions are completed. The VOF method was chosen for the model. The VOF model is a method proposed by Hirt and Nichols [20], which can be used to determine the interface between liquid and gas or two different liquids. The analysis was re-run by changing the discharge in the 'Boundary Conditions' section for each different discharge. In ANSYS-Fluent, measurements can be taken for many parameters such as current lines, water pressure, velocity, pressure. It also allows the output to be taken visually. The results and interpretations of the data obtained from the numerical examinations lasting approximately two to three days for each discharge and the readings obtained from the ADV are conveyed in detail in the research findings section.

## **RESULTS AND DISCUSSION**

In the experiments carried out in the hydraulic laboratory, the values read with ADV were recorded at 5 different points along the channel width expressing the z axis in the experimental setup where

the bridge piers positioned as a single axis were present. The modeling, which is the same as the channel dimensions, was done by installing ANSYS-Fluent. In Figure 3, it is seen that the comparison of the numerical velocity distributions obtained for four different discharge between the physical results and the numerical model results. It is seen that the ANSYS results obtained for the system positioned as a single axis are larger than the ADV results. However, it can be said that there is a significant similarity between the velocity distributions.

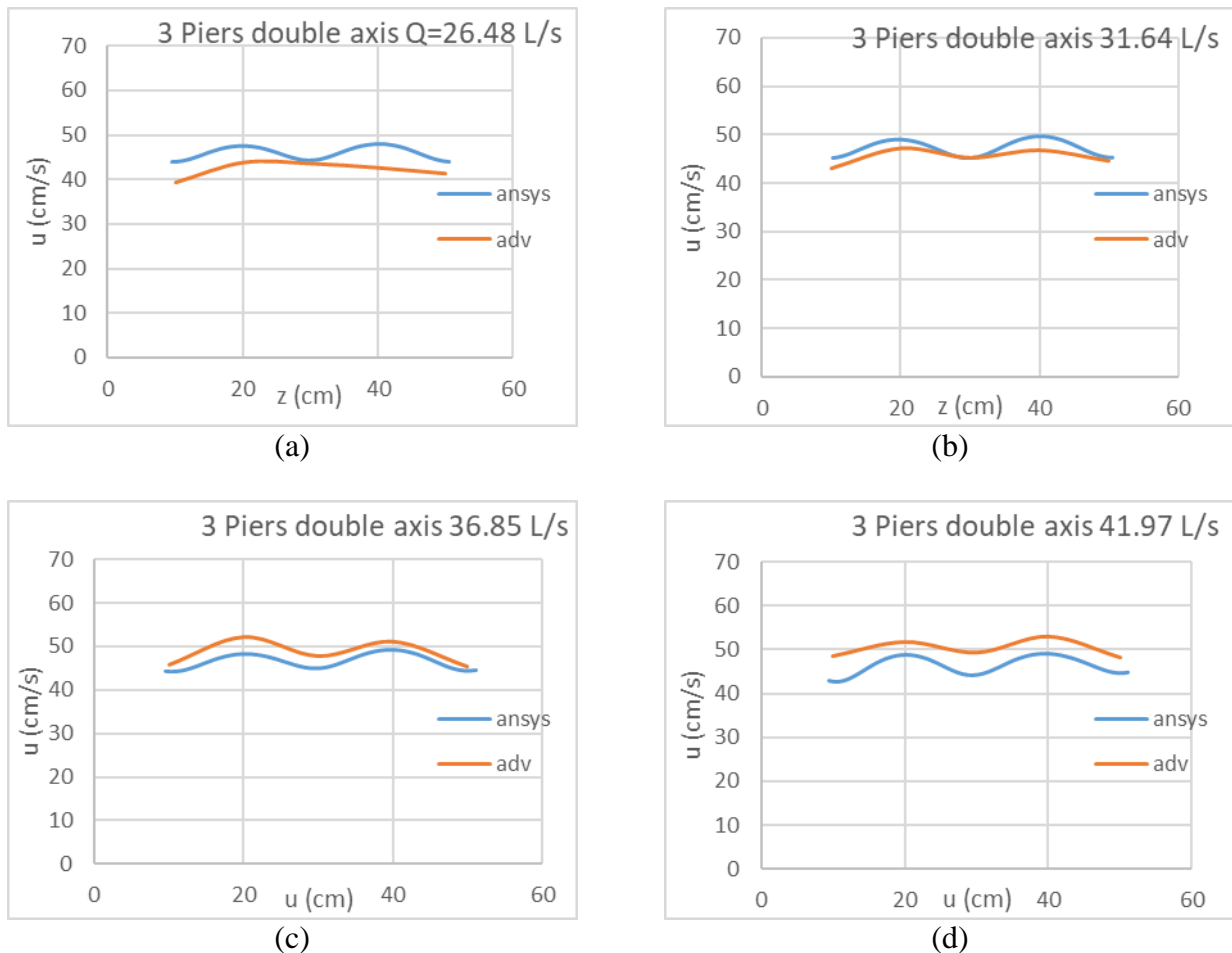


**Fig. 3.** Comparison of ANSYS and ADV for single axis:  
(a)  $Q= 26.55$  L/s; (b)  $Q= 31.73$  L/s; (c)  $Q= 36.98$  L/s; (d)  $Q= 42.19$  L/s

In the results obtained for the system positioned as a double axis, it is seen that the difference in ANSYS and ADV results is greater than other discharge due to the vortices in the system operated with a discharge of 31.64 L/s (Fig. 4). In general, it is seen that the numerical results and the physical results are compatible in this system.

The error rates between the ANSYS results and the ADV results were calculated using Eq. (1)

$$\% \text{ Mistake} = \left| \frac{WS_0 - WS_h}{WS_0} \right| * 100 \quad (1)$$



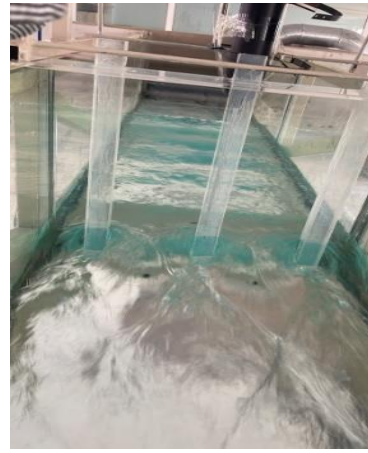
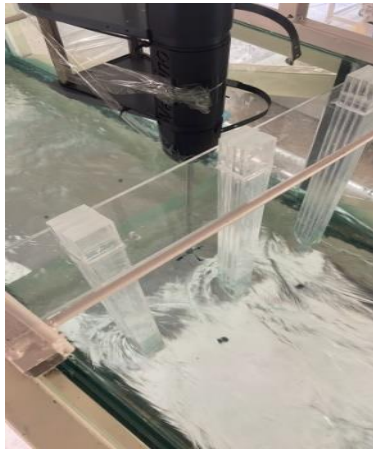
**Fig. 4.** Comparison of ANSYS and ADV for double axis:  
 (a)  $Q= 26.55$  L/s; (b)  $Q= 31.73$  L/s; (c)  $Q= 36.98$  L/s; (d)  $Q= 42.19$  L/s

Here, measured  $WS_{\bar{o}}$  is the physical result and  $WS_{\bar{n}}$  is the numerical result. Average error rates are given in Table 1.

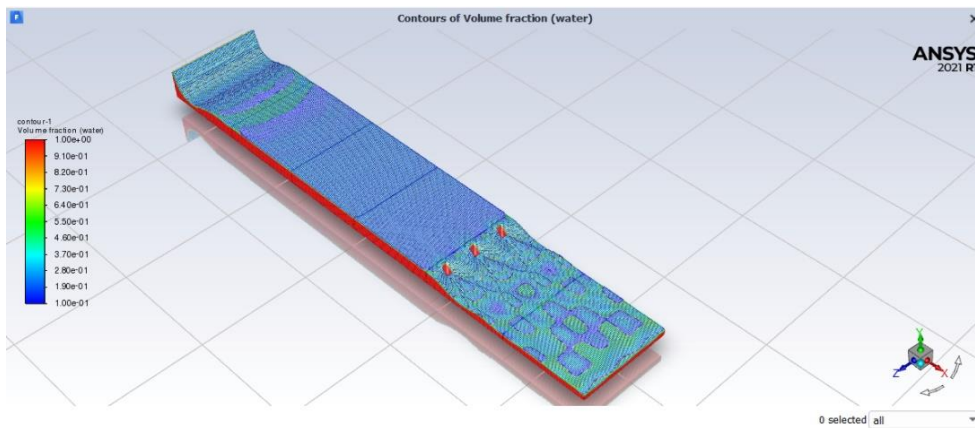
**Table 1.** The error rates between the ANSYS results and the ADV results

3 Piers Positioned as Single Axis				3 Piers Positioned as Double Axis			
26.55	31.73	36.98	42.19	26.48	31.64	36.85	41.97
L/s	L/s	L/s	L/s	L/s	L/s	L/s	L/s
2.854	5.661	11.007	15.935	11.640	5.059	3.407	11.651
7.536	8.201	12.764	16.911	8.581	3.598	7.666	5.851
8.625	11.635	11.343	15.536	1.806	0.044	5.871	10.417
3.499	4.065	8.428	10.271	12.475	5.925	3.855	7.524
1.302	2.131	9.136	10.101	6.649	1.637	2.066	7.414

Comparative representations of physical and numerical results are given in figure 5 for a discharge of 26.55 L/s taken at a distance of approximately 10 cm from the middle piers towards the upstream in the system where three square bridge piers are positioned as a single axis.



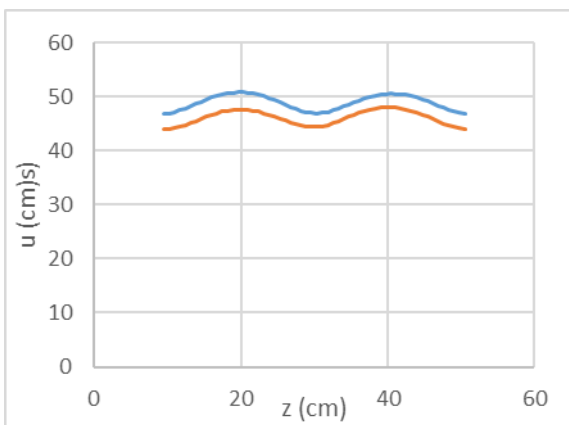
(a)



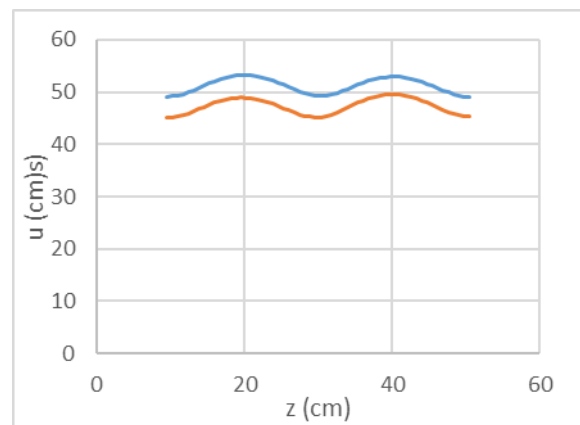
(b)

**Fig. 5.** 3 square bridge piers placing single axis:  
 (a) Experimental placing (b) Numerical placing

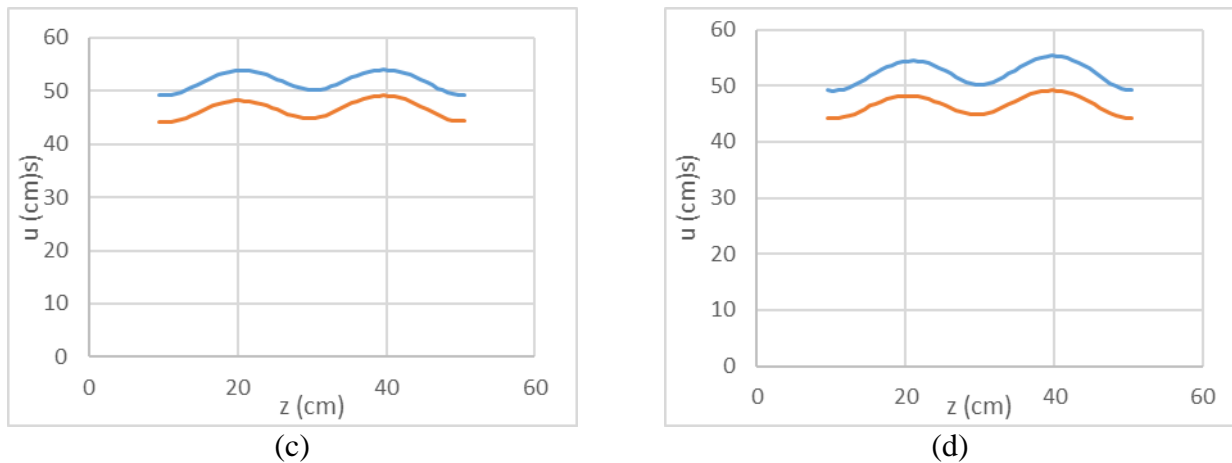
When the results obtained from ANSYS are compared as single axis and double axis systems, it is shown in figure 6 that the velocity distributions in the systems with bridge piers positioned as single axis are greater than the systems positioned as double axis. As a result of placing the bridge piers as double axis, it was determined that the flow velocity acting on the piers decreased. It is known that with the decrease in the flow velocity, the damage that may occur in the piers will decrease.



(a)



(b)



**Fig. 6.** Comparison of ANSYS results for single axis and double axis  
 (a)  $Q= 26.55$  L/s and  $26.48$  L/s (b)  $Q= 31.73$  L/s and  $31.64$  L/s  
 (c)  $Q= 36.98$  L/s and  $36.85$  L/s (d)  $Q= 42.19$  L/s and  $41.97$  L/s

## CONCLUSIONS

In this study, the velocity distributions that occur before the bridge piers in the systems placed as single axis and double axis with the same geometry were examined physically and numerically. The 3D model of the open channel system was created using the ANSYS-Fluent program and the results obtained from this program were compared with the ADV results used in the laboratory environment. As a result of the study, it was determined that the ANSYS used in determining the flow velocity area was very compatible with the physical results and that the flow velocity acting in the system obtained by placing the bridge piers as a single axis are larger. Increased flow velocity will affect the drag force of the flow; it will bring problems such as scouring and piling.

## ACKNOWLEDGEMENTS

This study is based on ongoing Ph.D. Thesis of Gökür Elif Yarbaşı Kayhanlar.

## REFERENCES

- [1] YANMAZ, A.M., Bridge Hydraulics, METU, Ankara. 2002, p. 5.
- [2] YURTSEVEN, M. L., Research of scour at the back of the Bridge Piers, Master Thesis, Istanbul Technical University, Institute of Science, Istanbul, 2005.
- [3] RICHARDSON, J.E., PANCHANG, V.G., J. Hydraul. Eng. ASCE, **124**, no. 5, 1998, p. 530.
- [4] ASHGRIZ, N., BARBAT, T., WANG, G., Int. J. Numer. Methods Fluids, **44**, no. 1, 2004, p. 1.
- [5] BARBHUIYA, A. K., DEY, S., Flow Meas. Instrum., **15**, no. 2, 2004, p.87.
- [6] GUMUS, V., SIMSEK, O., BAL, S., *Gazi Univ. J. Sci.*, **7**, no. 4, 2019, p. 938.
- [7] HUANG, W., YANG, Q. VE XIAO, H., Computers & Fluids, **38**, no. 5, 2009, p. 1050.
- [8] FAYYADH, M., AKIB, S., OTHMAN, I., RAZAK, H.A., Simul. Model Pract. Theory, **19**, no. 9, 2011, p. 1795.
- [9] SOYDAN, N. G., SIMSEK, O., AKOZ, M. S., Journal of Polytechnic, **21**, no. 1, 2018, p. 137.
- [10] ZHU, Z., LIU, Z., J. Cent. South Univ., **19**, 2012, p. 273.
- [11] AFZAL, M.S., BIHS, H., KAMATH, A., ARNTSEN, O.A., J. Offshore Mech. Arct. Eng., **137**, no. 3, 2015, <https://doi.org/10.1115/1.4029999>.
- [12] HAGER, M.W.H., SCHWALT, M., J. Irrig. Drain Eng., **120**, no. 1, 1994, p.13.
- [13] CHANSON, H., MONTES, J. S., J. Irrig. Drain Eng., **124**, no. 3, 1998, p. 152.
- [14] YILDIZ, M.C., YARAR, A., Konya J. Engine. Sci., **8**, no.1, 2020, p. 164.
- [15] FATHI-MOGHADDAM M., TAVAKOL M. T., RAHMANSHAHI M., Flow Meas. Instr., **62**, 2018, p. 93.
- [16] KOCAER, O., YARAR, A., Water Resour. Manag., **34**, 2020, p. 3949.
- [17] HAN S. S., CHEN Z., ZHOU F.Y., LU X. Q., Water Resour. Manag., **28**, 2014, p. 3111.
- [18] SONG C., ZHOU F., J. Hydraul. Eng. ASCE, **125**, no. 9, 1999 p. 959.

[19] MOUKALLED, F., MAGANI, L., DARWISH, M., The finite volume method in computational fluid dynamics, Springer International Publishing, Switzerland, 2016, p. 3-4.

[20] HIRT C. W., NICHOLS B.D., J. Comput. Phys., **39** no. 1, 1981, p. 201.

Citation: Kayhanlar, G.E.Y., Yazar, A., Experimental and numerical modelling of bridge piers having different number of axis r, *Rom. J. Ecol. Environ. Chem.*, **2023**, 5, no. 2, pp. 89-96.



© 2023 by the authors. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<http://creativecommons.Org/licenses/by/4.0/>).