International Journal of Engineering Research and General Science Volume 10, Issue 4, July-August 2022
ISSN 2091-2730

An Investigation of Water-Flow Pressure Distribution on Special Pier Shape in Iraqi bridge

Haıtham Fadhıl Shıhab, Prof. Dr. Ender SARIFAKIOĞLU, Salih Yilmaz
Cankiri Karatekin University, Civil Engineering Department, 18100, Cankiri, Turkiye
hythmalmshhdanya@gmail.com , enders@karatekin.edu.tr, salihyilmaz@karatekin.edu.tr

Abstract: - The special geometric of the viaduct pier that influences the constancy of the bridge knows the sum of backing throughout the overflow. The pier is considered the most important part of the bridge. The pier geometry is influencing all other bridge components. The shape of the geometry affects the forces on the pier due to water flow characteristics. Therefore, the main goal of this paper is to investigate the differences in pressure and water flow characteristics in a special pier shape in an Iraqi bridge. This goal was achieved by using the CFD ANSYS software program. The investigation was done using the Iron Bridge piers due to their unique shape. The study observed that with the increase of pier front face shape, there is a vortex will appear and the pressure will be high. The vortex line produced a score behind the pier which represents a big problem for the pier stability. To avoid this problem, the present study optimizes the pier design by adding a curve in the front face.

Keywords: pier shape, CFD, water flow characteristics

INTRODUCTION

The bridge design is usually associated with the idea of an open channel flow state; in addition, the flow mode can shift to water pressure flow if the lower edge of the bridge deck is totally or partially submerged throughout a major flood. Walls and floor effects of the river can be neglected in large river cross sections and bridge deck elevation. However, a combination of such geometry with submergences of the bridge deck is of limited practical interest. For the design and evaluation of river bridges, the most consideration occurs when the free surface prevents one from applying direct models of fluid structure interactions derived for unbounded domains. The reflection of the failure and damaged reflected on the repair cost[1]. It can be classified on direct cost related to repairing river bridges damaged by flood. Nevertheless, indirect cost considered due to business disruption and predicted to be exceeding five times the cost of direct repair. While design, planning, and building phases of the bridge are wisely calculated to achieve its application; piers repairs and protection from flow effect is given less importance than the structure will have. A maintenance strategy is required to avoid the problem of failure and the excess expenses during the life of the bridge. In order to develop a successful bridge, the successful plan must be taking in consideration established on the design procedure, involving the design method of bridges, maintenance program and construction methods. All variables will lead to a long life of structures. When the water flow affects the bridge pier, the pressure of the influx stream takes into account the fluid structure interaction. The dynamic interaction of structures that vibrate horizontally when in contact with one of two sides of water and the pressure effect of the hydrodynamic loads, vibration periods, and seismic reactions of structural water systems, including higher mode effects, occur. Assessing the consequences of the risk related with the possibility of water flow and the system of crossing the stream as a tool to develop site-specific design criteria. This assessment take into account capital cost, environmental hazards, and possessions influences on human life. The pier spacing and direction should be implemented by the hydraulic designer and the shape should be designed to minimize flow turbulence and possible impacts [2]. The foundations of the pier must be designed in such a way as to avoid damage due to wash out without preventive measures. Aghaee and Hakimzadeh made a simulation of turbulent flow around a circular pier based on the connection of velocity and pressure. Also, they investigated the nearby the pier vortexes by applying the calculation of Reynolds-averaged Navier-Stokes and by using the Navier-Stokes space-averaged calculations. Few systematic studies have been carried out on the pressure of the water flow on the pier taking into account the effect of the impact. The fluid-structure interaction on piers considered a crucial issue in this studies[3]. The water flow affects the bridge pier and the impact process around the pier caused by the flow can be categorized into two parts: the moment of impact on the bridge pier and the flow of water around the pier after the impact moment. Therefore, the flow influences on pier, the moment impact effect, as “water hammer effect,” which results a fluid pressure impact with larger magnitude than the effect of static fluid pressure which produced on the pier.

OVERFLOW EFFECTS ON BRIDGE PIER

When the overflow affects the bridge pier, the dynamic restriction and outgoing current pressure while taking the fluid structure interaction into account. Miguel and Bouaanani proposed a useful formulation for research into the dynamic constraint of a frame that vibrates laterally in contact with water on one or both parts and improves simplified procedures for the practical evaluation of vibration intervals, hydrodynamic loads, and seismic constraints of structural water systems including effects of the higher method [4]. However, relatively few systematic studies have been performed on a water flow pressure pier considering the effects, especially with respect to fluid structure interactions. When the overflow affects the bridge pier, the impact process caused by the outflow on the pier can be divided into two parts: the instantaneous effect of the flood on the bridge pier and the movement that flows around the pier after the instantaneous impact. The moment effect of the connected fluid structure system. The maximum displacement is sought at the higher end of the bridge and the extreme stress in the lower section of the bridge [1]. The effectiveness of Fluid-Structure is another important issue in this field. Efficiency of fluid structure on water pressure after the effect of the moment, effect on the water flow pressure used for the bridge is coupled when the overflow affects the bridge in a steady state, the maximum displacement at the higher end of the bridge pier and the pressure extremes in the lower section of the bridge pier is considered the amount studied [5]. Many researchers have studied the structure of bridges under flood loading. The design of river bridges has different sides; some researchers consider the effect of structural behavior and focus on structural modelling. On the other hand some of them have focused on the dynamic aspects of water flow, containing drag forces, various parameters for fluids such as Reynolds numbers, drag coefficients, and various procedures for simulation modeling. The behavior of fluids is complex and fluid structure interaction (FSI) is a aspect that has been investigated by different scholars [6][1].Variation of the fluid structure coupling effect coefficient with the current to the outflow speed, where FSI is known as the ratio of the numerical constraint of the bridge under the pressure of the water flow taking in consideration the effect of the fluid structure coupling when the overflow affects the bridge pier in a steady state, to the restriction of the bridge pier under pressure water flow except for the fluid structure coupling effect.

SWIRL AROUND BRIDGE PIER

The swirl is the basic mechanism command to local scour at piers or abutments. The accumulation of water on the upstream visage and Subsequent velocity of the flow directs the vortex around the front of the pier or pillar. A score hole improves when the mean transport of sediment away from the local area is more than the average transport of sediment away from the area. As the profundity of scouring raise, the power of the swirl is reduced, thus, diminishing the transport average. As equilibrium is reestablished, garbage ceases and the scour hole will not increase further. A typical swirl around a pier is: 1- Horseshoe swirl 2- Wake swirl. 3- Bow wave 4- Trailing Swirl These vortices are accountable or creating holes close to piers. This kind of corrosion is damaging many bridges (Hamidi and Siadatmousavi 2017). The horseshoe swirl is the outcome of 3D separation of the boundary layer that coils upward in a vortex forward and alongside parts of the cylinder pier (Melville, 1975). Part of the outflow approaching piers is deflected manner to the bed and twists up to create what is often depicted as a "horseshoe swirl" around the front visage of the structure. This swirly s often referred to as horseshoe swirl because of its great propinquity to a horseshoe. Shen et al. (2019) depicts the horseshoe swirls system in detail. Also, the separation of the outflow at piers' parts produces the so called wake-vortices [7]. Wake vortices are transferred downstream by the approach outflow and acts rather similar to a vacuum cleaner when it removes the downward material, which is then conveyed through the downward and horseshoe vortex (Melville, 2017). The wake- swirl system consists of unstable shear layers formed during pier smoothing and is separated from any part of the pier by a separation strip [8]. Wake swirl and the horseshoe wear down residue from the pier base. In numerous cases, materials which are removed by these swirl are redeposited promptly downstream. The power of the vortices is reduced with distance downstream command to a common sediment deposition downstream of piers[3][8].

FACTORS AFFECTING LOCAL SCOUR

The follow parameters have been recognized as factors influencing the profundity of scour thereabout a pier:

i. Width of the pier

The width of piers has a straight influence on the profundity of the scour. With the piers' width, the speed of outflow in the bridge opening raise to maintain continuity and leads to safety in scour profundity (Richardson, 2001). The effectiveness of pier dimension has been studied by Shen et al., (1969), Breusers et al., (1977). These researchers find out that horseshoe swirl form and power are the leading reason of garbage which are functions of piers' size. The bigger the size of the pier the greater the depth of the clear pit and the length of season it takes for it to develop a certain shear pressure ratio.

ii. Effect of Outflow Intensity

Outflow intensity is known as the ratio of the shear speed to the climacteric shear speed or the proportion of the mean speed of approach outflow (V) to the climacteric mean speed (Melville and Sutherland, 2018). Under clear water supply, the depth of local cleanup in uniform sediments rises almost linearly with outflow intensity to maximum velocity at minimum (Melville and Chiel, 2019). The extreme scour profundity is reached when the proportion Vs/V*c equals to 1 or V/Vc equals 1. The corresponding extreme scour profundity is called the starting peak. no clear occurs when outflow intensity V/Vc < 0.5, while clear water scour provisions happens for both orderly and no uniform sediments when the flow intensity ranges 0.5 < VNc < 1.0. The scour profundity rises almost linearly with V, and if VNc > 1.0 (live-bed clear) or clear with sediment movement. When the velocity outrun the starting speed, the local scour depth in the organized sediment decreases first and then rises again to a second highest level. These changes are relatively small as the high threshold is not fatigued and the sediments are structured. The second rise occurs in the flat transitional phase of the sediment transported on the channel layer and is called the live-bed high. The variable scour depth with outflow density is explained in terms of the balance between sediment input and output from the clear pit. The general conclusion was that the extreme local scour profundity in orderly sediments occurs at the threshold provision for clear-water scour provisions[9][10].

iii. Effect of Outflow

The depth over the apparent depth of flow is usually referred to as the shallowness of the outflow and is checked by relating the flow profundity (y) to the width of the pier (b). The effect of water profundity on clear is a controversial subject. Etta (2020) showed that, for shallow flows, a flattening cylinder (or arc wave) forms in front of the bridge pier which interferes with the apparent action of the horseshoe vortex due to the reverse senses of revolution. When the depth of flow rises, the interference of the flatness roller with the down- outflow and horseshoe swirly reduced and the effect of flow profundity becomes insignificant.

iv. Effect of Pier shape

The geometry of the pier is one of the primary factors that act an important role in the creation and strength of the circulation system. Bridge piers are constructed in various sizes and shapes. The most known shapes used are rectangular, square, round, oblong with beveled ends, oval, rectangular. The shape effect was calculated by several scholars who classified pier shapes into two categories, sharp-nosed piers and blunt-nosed piers. The blunt-nosed piers form a powerful horseshoe-shaped vortex system; therefore, the great depth of scour happens in the nose of the pier. The shape of the upper pier shall have strong efficacy on the depth of the scour and the length of the pier, the shape of the lower pier shall have an effect on the floor If the blunt-nose pier is in line with the outflow, where the horseshoe vortex system is very weak and very deep scour occurs near the estuary end. The geometry of piers is gate compacting width, length, shape and the alignment with outflow (Melville, 2008). Piers types can be divided to two kinds simple piers and complicated piers. The first type is simple piers which are the piers a containing fixed section throughout their depth. The second type is complicated piers which are the piers having pile foundations, caissons, slab foundations and tapered piers. The shape of piers greatly affected the amount of clear. With a pier, streamlining the front end reduces the power of the horseshoe whirl and reduces clear depth. Streamlining the downstream end of piers reduces the power of the wake vortices. Shape and alignment effects on local clear are given as multiplying correction factors. By investigating, form factors are important only if it is possible to ensure axial outflow. Even a small impact angle cancels out the benefits of the pier form.

4. experimental results and discussion

The tests designed to apply different water velocity values that can reflect the real conditions. NSYS CFD Fluent software used in run the experiments which contains two groups of scenarios. The experimentations are designed because of two different water velocities. The specific speeds depend on three concerns; the first concern is velocities (0.2, 0.3 and 0.5) meter per second which is used as river water flow speed. All these speeds applied on the brides in Iraq subjected on Degla river, Baghdad Iraq named the Iron Bridge and Al-Muadam Bridge. The results of the CFD simulation for the three kinds of pier analyzed the properties of the water flow and evaluated the pressure applied to it based on different levels in (x, y and z) directions and directed the scholar to the optimal outcomes for comparison. Current outcomes achieve the purpose of this study and provide a brief discussion based on four sections. The first section presents the aerodynamic analysis of the iron bridge by means of ANSYS outcomes and plotting the numerical outcomes. The second division offers the aerodynamic analysis of Al-Muadam Bridge. At the end an optimization presented with a comparison.

The water flow forces on the bridge piers are very complex function of the pier shape. The amount of water flow forces generated by an object depends on the flow behavior on the shape of the object. For that, optimizing the shape represents the crucial case in pier design[11][12]. The complex dependencies of pier shape, flow conditions and inclinations can be expressed by the water flow conditions which they considered as the hydrodynamic loads to model all the complex dependencies[6]. Therefore, the standard approach for pier design optimization depends on the flow conditions of the hydrodynamic model[2]. These forces concern about the working parameters in the pier which represent quantities that can be considered a risk assessment on the bridge pier. The dramatic influence on the construction structure of the pier and bridge affects the total construction. The reason for this phenomenon is the hydrodynamic characteristics change on the pier which is determined from the shape of the pier design. The main problem is how to decide which shape of the piers is suitable to reduce the influence of water stream and fluid-structure interaction on the bridge pier. It is a major challenge due to reflection of many flow effects and behavior round the pier correlated with the fluid-structure coupling influence.

FLUENT MODELS

Optimization of the CID perturbation pattern is required to select the appropriate undoing method. Advances are urgently needed in the prediction of turbulent outflow flux with strong reversible pressure gradients and the interaction of separation with boundary layer outflow. In computational methods, the Reynolds average (RANS) methodology is the most widely employed practical method due to its reliability and height accuracy, and the low computational fluid dynamics (CFD) method. [13].

By making use of the momentum equation, the total traction force can be found. A plentiful number of studies have been made regarding the properties of traction and bridge piers. Holley (2001) conducted a lag study on bridge piers and traction properties. Traction properties of different cladding frame geometries with remission rates of 0.33 and 0.4 for critical outflow provisions were studied [14]. Agarwal et al. (2014) carried out an experimental study on the traction properties of cylindrical pillars with various gap collar derivatives. Several studies are directly related to the scope of our study and are briefly summarized below. Holley (2001) carried out an experiment the effects of standing water, traction forces, and changes in water parameters on both the top and course of bridge piers. In this experimental work, various sizes of artificial bridge piers in the laboratory under the effect of the critical outflow. The study cleared that the stagnant water equation can be used to calculate the water standard. The discharge changes the standard of standing water in the upper part. Also The experimental results with the y formula was compared [15]. The specimen used RANS equations and K-"perturbation sample. The numerical study is based on experimental laboratory tests, the study also attempts to apply the transient testing phase of all outflow fields around the pier pressure and bed shear [2][16]. In the study of Laugot, et al. (2010), the purpose of the study was to take a sample from the scour hole of the turbulent outflows of the open channel through the overflow. The 2D channel is artificially induced under sub-climate outflow conditions with Reynolds and Freud numbers in the ranges between 5,000 and 10,000, correspondingly[14][17]. Numerical results of the outflow are calculated using standard turbulent samples k- "and RNG k-" and the dimensions of the free surface is studied with the VOF sample given in the Fluent code[15][18]. The objective was to study the interaction between the free plain and the wave to improve the separation points. Both the conventional k-co specimen and the transformed k-specimen are multiplied by the blending function and both specimens are added together. The blending function is defined as one in the wall range that activates the standard k-co pattern and zero away from the plain that activates the transformed k-pattern. In addition, the SST specimen contains the expression derivative attenuated by diffusion in the joint equation[19]. Scholars are trying to improve these methods by modifying the turbulent speed to calculate the turbulent shear stress transfer. In 1994, Meat enhanced the SST (Shear Stress Transfer) sample based on k-co. Uses the k-co specimen around flatness and the k-e specimen in the free shear layers. The advances mentioned make the SST k-co specimen further accurate and dependable for a larger class of outflows than the standard k-co specimen. For these causes and the moderately high efficiency of the numerical solution, the two-equation SST k-co specimen is set as a separate technique for the body-flow numerical simulation [16]. Therefore, this technique was employed in this study because the properties of combining both the standard k-co specimen and the features of the standard k-a model. In addition to that, the specimen equations act appropriately in both the wall and far regions.

FLOW AROUND THE BRIDGE PIERS PLANE

Current study, resulted data were obtained on the horizontal planes of the bridge piers. The outcomes on the field of flow and turbulence features of the flow around bridge piers and the influence of spacing among columns are offered in the succeeding sections. The pier is the most crucial component in the bridge. For that, this study aims to investigate the aerodynamic characteristic in bridge pier. The flow characteristic can be recognized by observing the water flow behavior in different types of piers and water velocity. Different shape of the bridge piers produces different flow behavior around the two sides of the bridge pier. This generates a compression change founded on a physical rule identified as the Bernoulli principle. As this compression attempts to equilibrate, the water flow tends to move in the direction of low compression. In order to characterize the aerodynamic forces acting on the bridge pier. The present study observes the aerodynamic behavior around the bridge pier by using CFD

ANSYS role of CFDs in design changes rely on the phase of the study. Computational fluid dynamics are used in the shape of a two-dimensional, structured lattice flow solver with mutually Euler and Reynolds mean Navier-Stokes equations. The numerical model is prepared and run with CAD software that creates boundary conditions for the analysis. This analysis outcomes act a significant part in optimizing the design mechanisms so that there is slightest pressure on the pier surface and swirl of water flow.

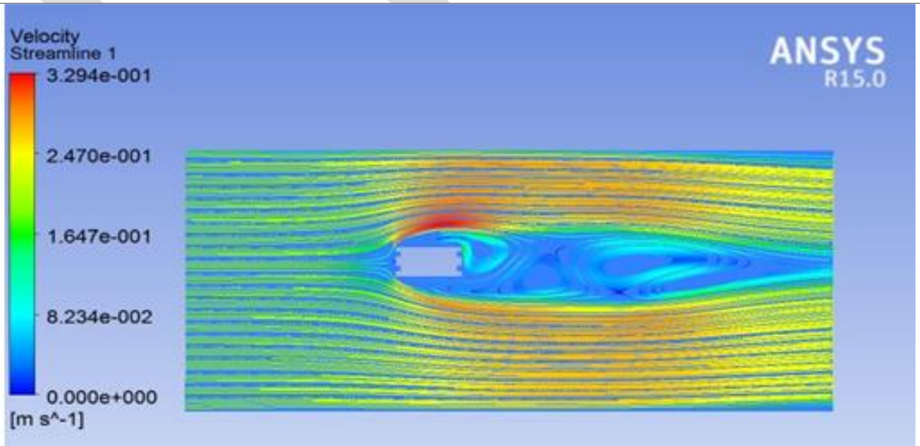
6.1 Iron Bridge Flow Pattern

The vector diagrams and flow line diagrams as revealed in Figure 4.2 for the case of a single column obviously represent the flow patterns around the bridge pier. ANSYS simulates the chosen cases by defining them according to boundary conditions. Five basic properties used in the CFD-ANSYS simulation process are shown in Table 4.1. Various scenarios are produced using the pressure and velocity behavior of different bridge piers subjected to water motion.

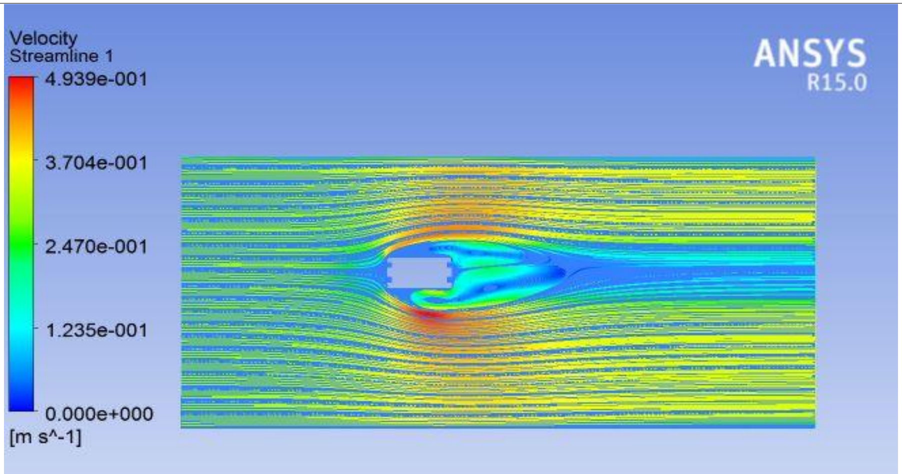
Table 4.1: Operating Parameters

	parameter	**input value**
1	Water flow speed	0.2, 0.3, 0.5 mps
2	Operating temperature	288.16 K
3	Density	1016 Kg/m3
4	Model	SST k-ω
5	Fluid	Water

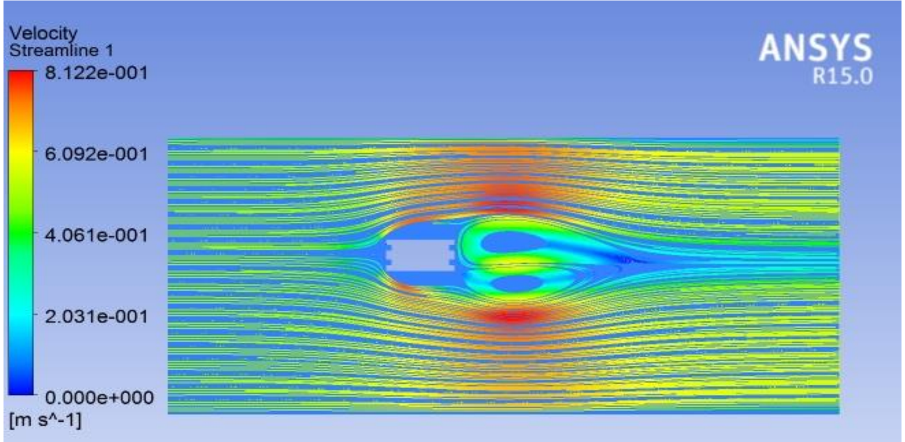
The flow characteristic can be recognized by observing the water flow behavior in different types of piers. The effect of pressure variation on the bridge pier indicate the differences. The water flow speed is crucial because it control the effect of water movement which changes the surface pressure on piers. It is observed that the maximum pressure affect in the tip of the bridge pier as shown in the red color of the pressure profile. This observation approved that there is a high down force in this point, the affected force component will have applied perpendicularly on the longitudinal axis of the bridge pier section. But with the increase of water velocity, the vortex appears as shown in Figure 1in the velocity profile. The vortex will increase the drag force which cause several aerodynamic problems. Due to boundary layer separation at the vacuum side of the bridge pier stationed below the critical edge vortex which broke down that resulted in loss of pressure.



Velocity Flow Pattern of 0.2 mps



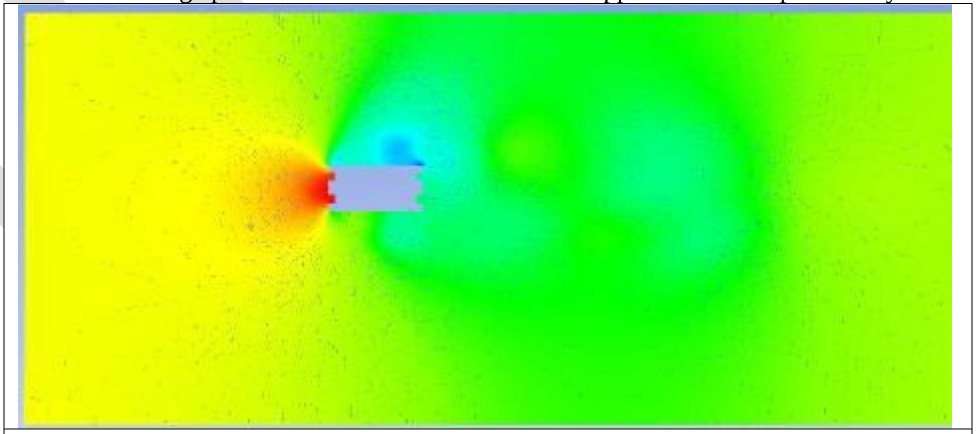
Velocity Flow Pattern of 0.4 mps



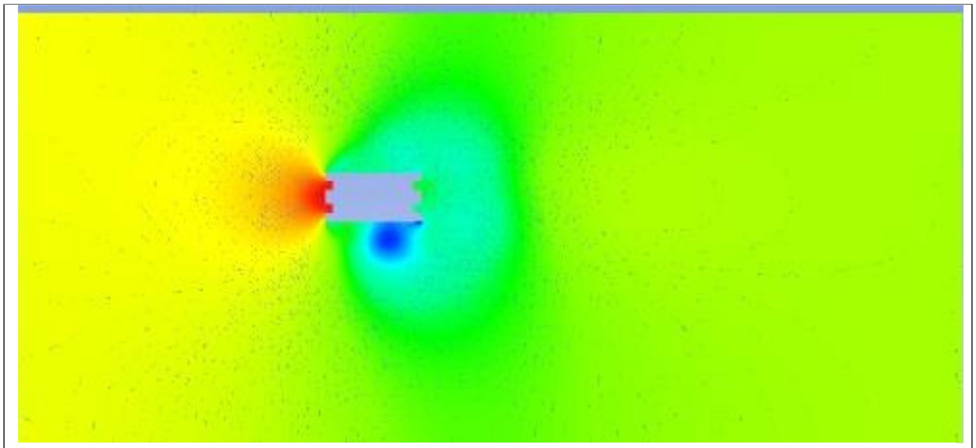
Velocity Flow Pattern of 0.5 mps

Figure 1: Velocity flow pattern of the iron bridge simulation results,

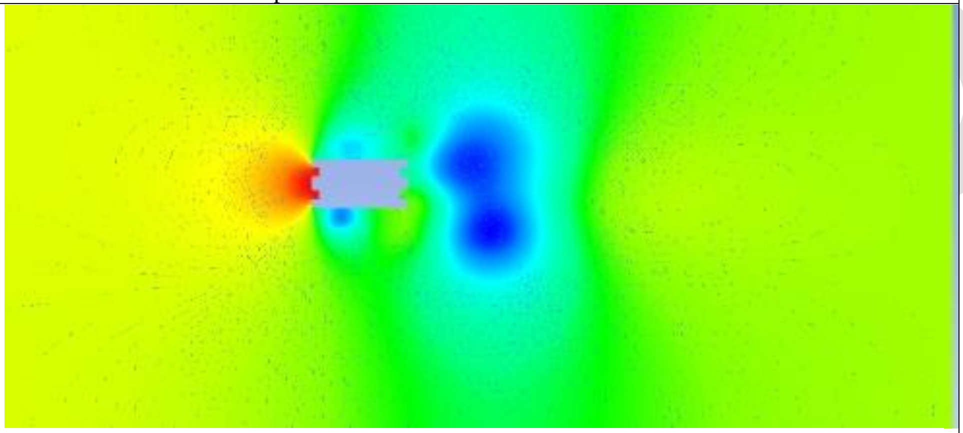
It is seen that the water characteristics have been changed by the changes of water velocity; therefore, the speed profiles observe a rise in water velocity at the side of the bridge pier surfaces and a reduction in the upper surface as specified by the red region.



Pressure Pattern of 0.2 mps



Pressure Pattern of 0.4 mps



Pressure Pattern of 0.5 mps

Figure 2: Pressure profile of the iron bridge simulation results

Pressure lines notice an rise in the front surface of the bridge pier and a reduction in the rear surface. This outcome specifies that the resistance against the direction of flow will create a negative force which means that the force will increase because there is a positive relationship between the force and the water pressure in the bridge pier surfaces.

VELOCITY NUMERICAL RESULTS OF THE IRON BRIDGE

The numerical results which obtained by the ANSYS -FLUNT processes present a set of values. These values represent the changes results of the velocity based on the changes in distances far away from the center of column. Figure 3 shows the distribution of water velocity distribution alongside three diverse longitudinal axes Y/D = 0, 1 and 2. Plans show pockets of greater and lesser speed values at the side and heels of the shaft, in that order. The rise velocity value on the column side is due to the blockage due to the presence of the column which reduced the flow. It is seen from the results that the velocity increases incrementally with the increasing of the Y/D. This fact can be observed clearly by plotting the results. Figure 4.5 to 7 shows the relationship of the bridge pier down force effect due to the shape effect.

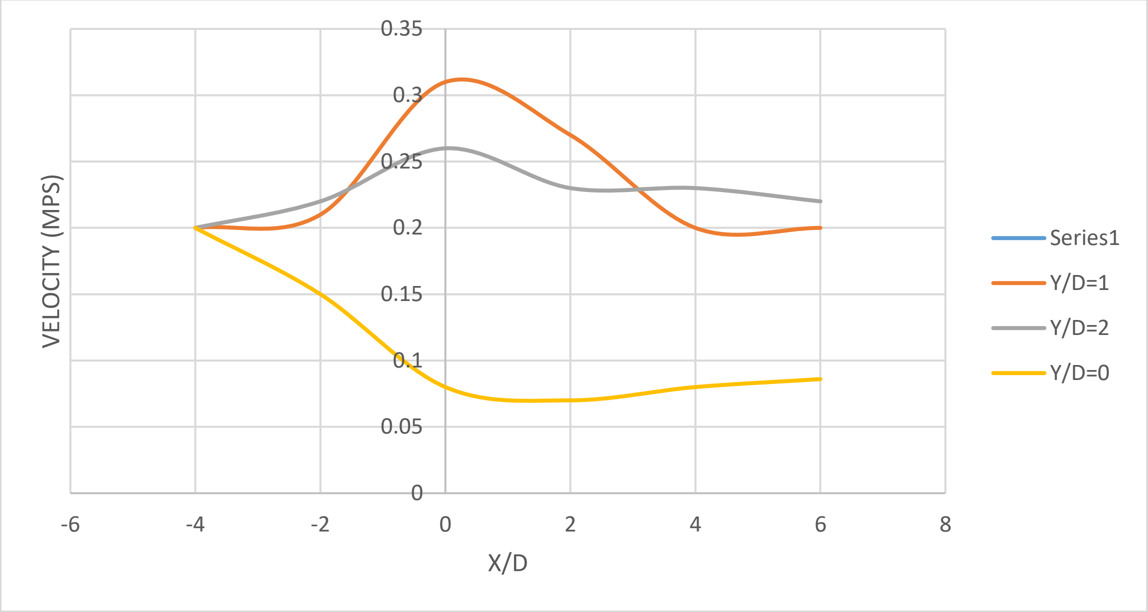


Figure 3: Cutting profile for single column case speed wise component of Iron Bridge, Vel. 0.2m/s

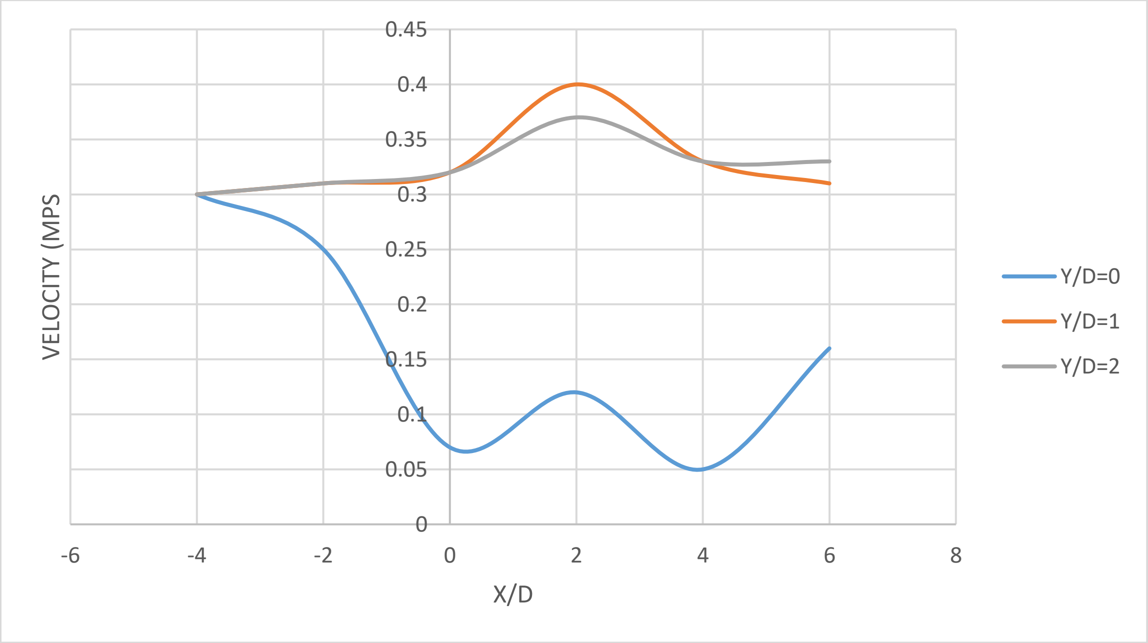


Figure 4: Cutting profile for single column case speed wise component of Iron Bridge, Vel. 0.3 m/s

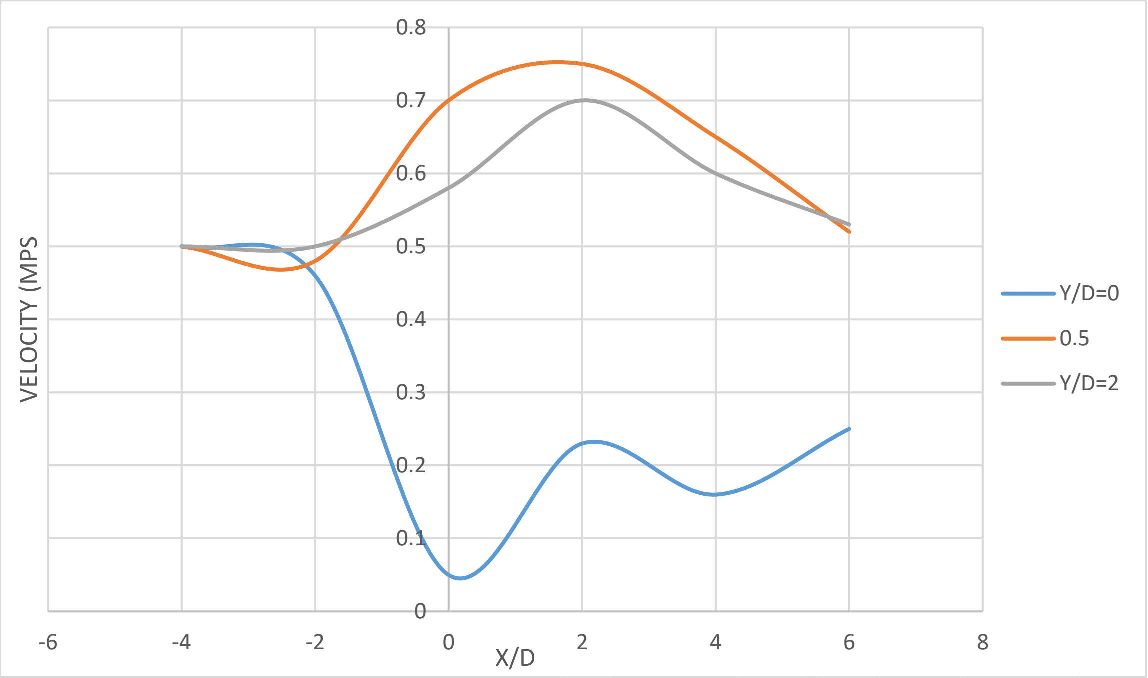


Figure 5: Cutting profile for single column case speed wise component of Iron Bridge, Vel. 0.5 m/s

PRESSURE NUMERICAL RESULTS OF THE IRON BRIDGE

Investigation of Aerodynamic characteristics achieved by running the ANSYS solver based on first case of bridge pier shape. It is detected that the compression distribution around the pier is convergent in both sides of the pier. The maximum pressure affect in the tip of the pier, this notification approves that the shape of the tip is an important issue, and the affected force component couldn't have applied perpendicularly on the longitudinal axis of the pier section.

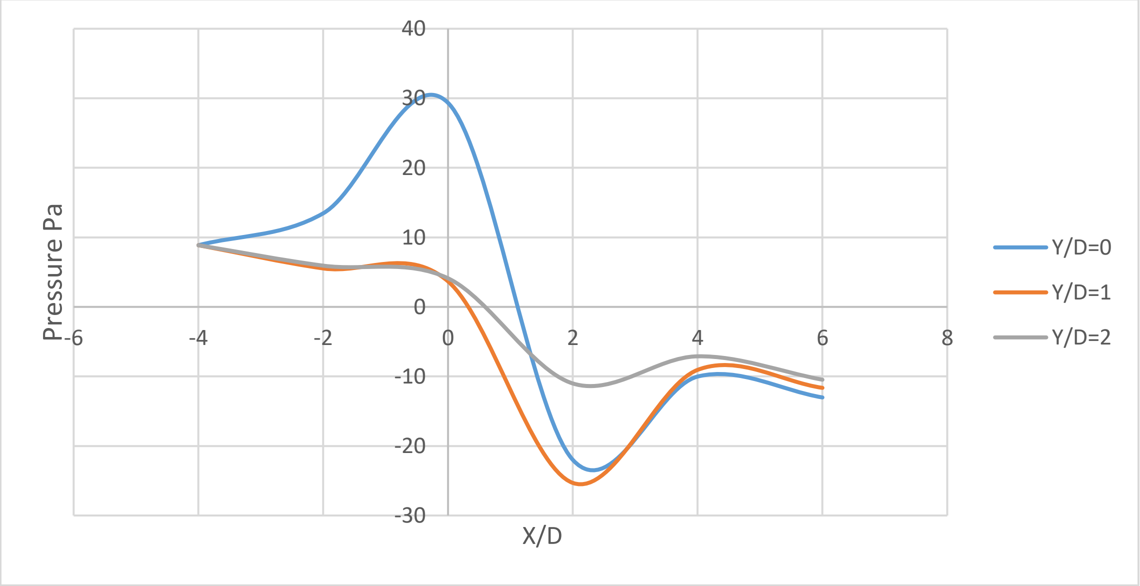


Figure 6: Cutting profile of compression component for single column case of Iron Bridge, Vel. 0.2 m/s

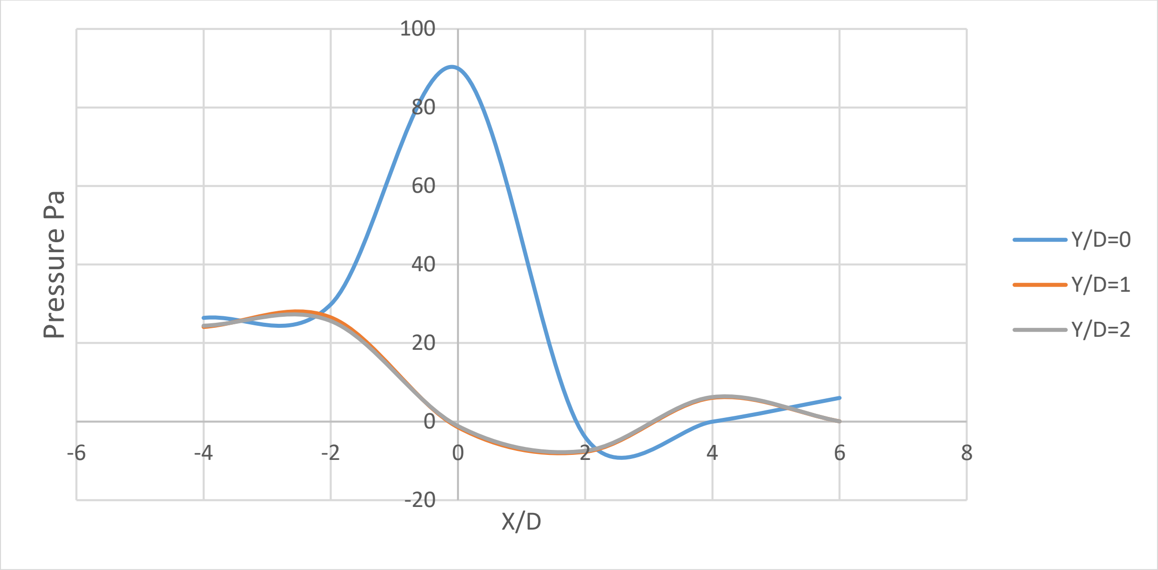


Figure 7: Cutting profile of compression component for single column case of Iron Bridge, Vel. 0.3 m/s

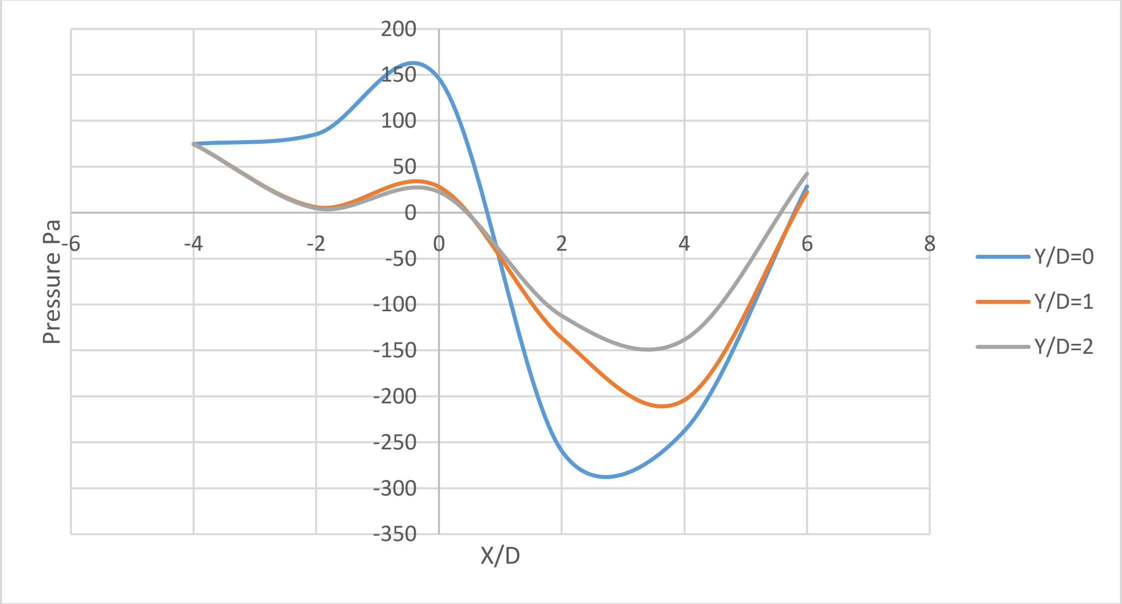


Figure 8: Cutting profile of compression component for single column case of Iron Bridge, Vel. 0.5 mps

OPTIMIZATION OF PIER FRONT FACE

Based on the present cases, the researcher develops a new front pier shape using circular shape. The finding of optimal pier founded on the existing simulations in earlier mentioned sections. Understanding the water flow behavior was useful to develop the front pier in order to get better understanding for pressure effect. Using front pier design provides unique results. Figure 4.20 shows the new suggestion design of the optimized pier.

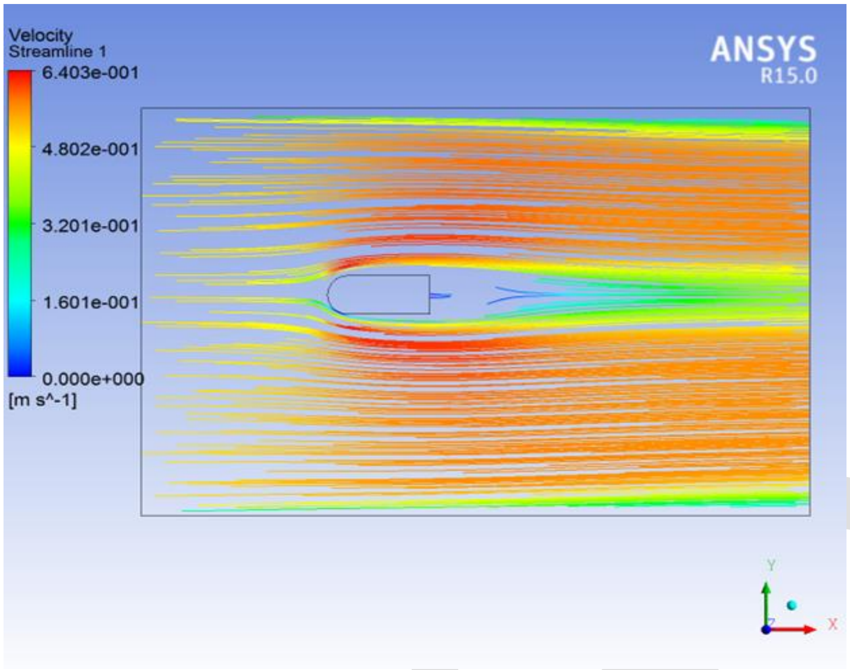


Figure 9: The New Suggested Front Pier Design (Velocity, 5 mps)

This suggested design gives a narrow front area to create minimum pressure load. And the curve in front of pier will provide less vortex effect. The researcher found from the presented work that there is an optimal shape of pier which can provide the best water flow.

CONCLUSION AND RECOMMENDATION

The CFD simulation results observe different turbulence models. The study observes that the pier design has a major effect on pressure distribution. In order to investigate the complex pier correlations, the forces are considered as a complex dependence of the shape, flow conditions, and also inclinations. The study observes that the optimal pressure distribution effect is concerned with the front curve which minimizes pressure load and reduces the vortex behind the pier. Based on these facts, the results of the piers test results summarized in the pier. The CFD ANSYS results observed that the pressure force increases incrementally on the pier. This result observes the importance of the surface shape of the pier to generate the pressure force based on Y/L AND X/D. The characteristics of the water surface layers around the pier surface affect the water 'movement behavior which is changed due to the changes in the pier shape. Also, the pressure and velocity values growth in polynomial progressive for each level Also, the water flow on the pier surface causes vortex lines leading from the pier tip.

REFERENCES:

[1] Y. H. Wang, Y. S. Zou, L. Q. Xu, and Z. Luo, "Analysis of water flow pressure on bridge piers considering the impact effect," *Math. Probl. Eng.*, vol. 2015, 2015, doi: 10.1155/2015/687535.

[2] H. K. Jalal, "Numerical Study of Optimum Pier Shape for Safe Bridge," no. September, 2019.

[3] Y. Aghaee and H. Hakimzadeh, "Three dimensional numerical modeling of flow around bridge piers using LES and RANS," *Int. Cconference Fluv. Hhydraulics, River Flow 2010*, no. 2000, pp. 211–218, 2010.

[4] N. Yousif *et al.*, "Fuid flow and fluid force acting on a cylindrical pier," *J. Phys. Ther. Sci.*, vol. 9, no. 1, pp. 1–11, 2018.

[5] F. Alijani and M. Amabili, "Non-linear vibrations of shells: A literature review from 2003 to 2013," *Int. J. Non. Linear. Mech.*, vol. 58, pp. 233–257, 2014, doi: 10.1016/j.ijnonlinmec.2013.09.012.

[6] M. Nasim, S. Setunge, S. Zhou, and H. Mohseni, "An investigation of water-flow pressure distribution on bridge piers under flood loading," *Struct. Infrastruct. Eng.*, vol. 15, no. 2, pp. 219–229, 2019, doi: 10.1080/15732479.2018.1545792.

[7] Y. Zhai, "DigitalCommons @ University of Nebraska - Lincoln TIME-DEPENDENT SCOUR DEPTH UNDER BRIDGE-SUBMERGED Presented to the Faculty of," 2010.

[8] T. Tsutsui, "Fluid flow and fluid force acting on a cylindrical pier standing in a scour," *Nihon Kikai Gakkai Ronbunshu, B Hen/Transactions Japan Soc. Mech. Eng. Part B*, vol. 73, no. 5, pp. 1191–1197, 2007, doi: 10.1299/kikaib.73.1191.

[9] Ł. Topczewski, J. Cieśla, P. Mikołajewski, P. Adamski, and Z. Markowski, "Monitoring of Scour Around Bridge Piers and Abutments," *Transp. Res. Procedia*, vol. 14, pp. 3963–3971, 2016, doi: 10.1016/j.trpro.2016.05.493.

[10] B. Melville, "The physics of local scour at bridge piers," *Proc. 4th Int. Conf. Scour Eros.*, no. 1, pp. 28–40, 2008.

[11] Z. Chen, L. Liu, Y. Hu, N. Ye, X. Shen, and Y. Wang, "Experimental on the protective performance of column bridge pier anti-scouring device," *E3S Web Conf.*, vol. 248, 2021, doi: 10.1051/e3sconf/202124801015.

[12] I. Carnacina, N. Leonardi, and S. Pagliara, "Characteristics of flow structure around cylindrical bridge piers in pressure-flow conditions," *Water (Switzerland)*, vol. 11, no. 11, 2019, doi: 10.3390/w11112240.

[13] K. R. Atkins, "Investigation Into the Flow Behaviour of a Naca 0021 Airfoil With Leading Edge Undulations Simulated With Rans in Star-Ccm+," no. September, p. 79, 2015.

[14] C. R. Suribabu, R. M. Sabarish, R. Narasimhan, and A. R. Chandhru, "Backwater Rise and Drag Characteristics of Bridge Piers under Sub- critical Flow Conditions," *Eur. Water*, vol. 36, no. 2010, pp. 27–35, 2011.

[15] P. Ipa and S. Di, "Viscous fluid flow," no. January, 2017.

[16] A. K. Saraf, M. P. Singh, and T. Chouhan, "A Review on Aerodynamic Behavior of Airfoil when Surface Modified," vol. 7, no. 3, pp. 516–519, 2016.

[17] S. Malavasi and A. Guadagnini, "Hydrodynamic Loading on River Bridges," *J. Hydraul. Eng.*, vol. 129, no. 11, pp. 854–861, 2003, doi: 10.1061/(asce)0733-9429(2003)129:11(854).

[18] Stanford University, "Simulation of Turbulent Flows, ME469B: Computational Methods in Fluid Dynamics Using Commercial CFD Codes," *Standford Univ.*, p. 69, 2004.

[19] H. SOĞUKPINAR, "Uçak Kanatlarinda Enİdeal Hücum Açisini Bulmaİçin 4 Rakamli Naca 00Xx KanProfillerininNümeriAnalizi," *Uludağ Univ. J. Fac. Eng.*, vol. 22, no. 1, pp. 169–169, 2017, doi: 10.17482/uumfd.309470.