

# Design improvement of a side water intake on a small hydropower plant

Miloš Nikolić<sup>1\*</sup>, Vladan Karamarković<sup>1</sup>, Rade Karamarković<sup>1</sup>, Miljan Marašević<sup>1</sup>

<sup>1</sup>Faculty of Mechanical and Civil Engineering in Kraljevo, University of Kragujevac, Kraljevo (Serbia)

*This paper deals with the problem of flow distribution at the water intake of a small hydropower plant (SHPP). The plant, with the capacity of 1.25 MW, has been in operation since the end of 2013. At the side intake of the SHPP an uneven distribution of water flow in three parallel chambers of a settling basin has been observed. This anomaly has been noticed during the rinse of different chambers of the settling basin. Each chamber precipitated different amounts of sand. In this paper, water intake is simulated using computational fluid dynamics (CFD) and a technical solution for equalizing flow in three chambers of the settling basin is recommended. The solution should be characterized by: minimal price, simplicity and the lowest possible pressure drop.*

**Keywords:** Small hydro, Water intake, CFD modelling

## 1. INTRODUCTION

Hydropower plants have a global installed capacity of 1064 GW in 2015 with overall productivity of the electric power of 3940 TWh, which represents about 16.6% of the total electricity production [1], [2]. On that basis, hydropower represents the largest source for the production of electricity from renewable energy sources.

Hydropower plants are usually classified based on the size or installed capacity (Table 1.).

Table 1. Classification of Hydropower by Size

Classification	Size
Micro hydropower	Up to 100 kW
Mini hydropower	Between 100 kW and 1000 kW
Small hydropower	Between 1 MW and 10 MW
Medium hydropower	Between 10 MW and 100 MW
Large hydropower	Larger than 100 MW

In Serbia, the energy potential of renewable energy sources is estimated at 4.8 Mtoe per year, of which 0.6 Mtoe is potential for hydropower [3], [4]. The Serbian hydropower potential for producing electricity is 17000 GWh, of which 10000 GWh was used, so that the remaining potential for use is 7000 GWh [5]. The potential of small hydro power plants (up to 10 MW) at 856 locations on rivers in Serbia, including small rivers, with installed capacity of 450 MW is 1,590 GWh per year [6], [7].

There are two basic types of hydro power plants: run of river and storage. Small hydro power plants are being built as a run of river power plant, in which the available flow of water diverted from rivers in the forebay, and the penstock leading to mechanical plant, which is located downstream.

Different types of water intake used in hydropower plants to divert water from the river. Run of river type of small hydro used three main types of water intake:

1) Side intake: The intake is located at the side of the river along one of the banks,

2) Frontal intake: The intake faces the river flow sometimes even perpendicular to the flow,

3) Bottom intake: The intake withdraws water from the riverbed and conveys it directly into the penstock.

In this paper is observed a small run of the river hydropower plant with a lateral water intake. The installed capacity of this power plant is 1.25 MW. In the exploitation conditions, it was observed that the abstraction of the water over the side water intake comes to uneven distribution of water flow in three parallel chambers of the settling basin, which leads to uneven distribution of sediment and create resistance that leads to a reduction of net head. The aim of this paper is to propose a technical solution for equalizing flow (velocity) of water in the chambers of the settling basin. The solution should be characterized by simplicity, minimum price and the lowest possible pressure drop. For the simulation and analysis of the proposed constructional changes at the water intake was used computational fluid dynamics (CFD). A two-dimensional CFD model of the side water intake of small hydropower plant has been presented in this paper for the simulation and analysis of the various structural changes in order to equalize the speed in the chambers of the settling basin. Results of a CFD model of the current situation are compared with different construction solutions in order to find the optimal solution of the perceived problem.

Similar CFD models are used in hydro Engineering for simulation of fluid flow and optimize the various structures for the abstraction or discharge of water. The two CFD models from [8] and [9] are interesting for the problem which is considered in this paper. In this study was used the commercially available CFD software package to simulate the flow of water in the water intake. The advantage of using the software packages (STAR-CD, FLUENT, CFX, or Flow3D) is that the codes are extensively tested and verified.

## 2. WATER INTAKE STRUCTURE

Planar structure model of the side water intake with the lake is shown in Figure 1. Maximum installed capacity of water intake amounts to 5.65 m<sup>3</sup>/s. A project is defined biologically required minimum flow rate of 740 l/s. The dam built across the width of the river is 14 m long.

\*Corresponding author: Faculty of Mechanical and Civil Engineering, University of Kragujevac, 36000 Kraljevo, Dositejeva 19 (Serbia), milosqw@gmail.com

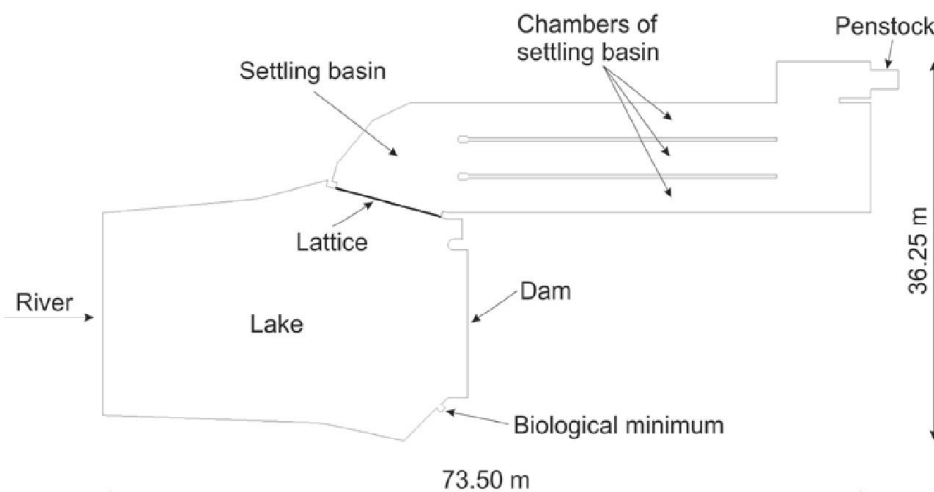


Figure 1: Water intake structure

In this type of hydropower plant and water intake has a role to ensure the projected water level. At the entrance to the settling basin is located lattice, 10.50 m wide and 1.20 m high which has a role to prevent large sediment or other elements that the river carries to enter to the settling basin. The width of the openings in the lattice is 25 mm, while the width of the solid material is 8 mm.

Sedimentation tank is 52.67 m long and 15.50 m wide, and consist of three parallel chambers which are 30.47 m long and 3.30 m wide. The chambers are divided by a concrete wall thickness of 0.30 m. The role of the settling basin in the water intake that directs water from the lake to the pipeline, ensuring the flow of pressure to mechanical plant, based on the projected level of water that held in the settling basin and the lake. The second function of the settling basin is to stop the smaller sediments that have passed through the lattice to avoid the pipeline and then to mechanical installations. For this reason settling basin consists of three chambers, of which is for rinse sediments in one chamber other two chambers closed by slide gate, and the total flow is directed to a chamber which is rinsing. In this way the velocity of the water increases significantly and water together with sediments get out from the settling basin into the river.

### 3. COMPUTATIONAL DOMAIN

#### 3.1. Computational grid

One of the most important steps in the use of commercial CFD software to the problem of fluid flow is to develop a computer grid with adequate resolution. In defining a grid, the essential is to find the optimal number of elements so that the model is credible in presenting the real state, and on the other side higher resolution form a larger number of elements and therefore these extend simulation time and achieving convergence [10].

Computational grid of the 2D water intake model which is observed in this paper consist of 217979 elements and 227339 nodes. In generating the grid using module Mesh in Ansys software, physical preferences are defined as CFD, solver preferences as Fluent, with relevance 100. In defining the size of the grid elements, it was selected a curvature size function with fine relevance. Curvature Normal Angle is set to 12.0 degrees. Minimum element size is 8.4 mm, while the maximum size of the area and max tet size are limited to 100 mm.

Because of the lattice which is relatively dense in relation to the defined grid, it is necessary in the field of lattice define local finer grid. Edge Sizing was used as a function with defined geometry of lattice edge. The parameter for generating local grid is the element size of a maximum of 5 mm.

#### 3.2. Boundary conditions

The boundary conditions defined on the two-dimensional CFD model of side water intake considered in the paper and shown in Figure 2. are: Fluid\_inlet, Fluid\_outlet, and Biological\_minimum.

Boundary condition at the entrance to forebay represent the Fluid\_inlet with the defined volumetric flow of water with a value of 6.39 m<sup>3</sup>/s. Velocity field of water at the entrance is normally distributed with respect to the boundary surface.

The model is defined that the output of the settling basin has set as a boundary condition "Outlet", Fluid\_outlet with the volumetric flow of water than 5.65 m<sup>3</sup>/s, which is the maximum installed flow of the small hydropower plant.

Biological minimum which is necessary to provide for the fish's path, is defined as a boundary condition "Outlet", with a water flow rate of 0.74 m<sup>3</sup>/s.

#### 3.3. Solving procedures

The parameter for comparison of the proposed technical solutions is the velocity of water flow in the chambers of the settling basin. To describe the fluid velocity and turbulence in the river or hydraulic systems, such as a water reservoir, water discharge structure, or water intake structure of small hydropower plants, are used Navier-Stokes equations. For turbulent flow are commonly used version of the Reynolds averaged Navier-Stokes equations (RANS). In this paper was used Pressure-Based RANS model, with absolutely defined speeds. It was suggested the planar, no gravity, and a stationary velocity field. RANS model defined in the [11] is used in this study for modeling fluid flow in the structure of water intake:

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = \frac{1}{\rho} \frac{\partial}{\partial x_j} (-P \delta_{ij} - \rho \overline{u_i u_j}) \quad (1)$$

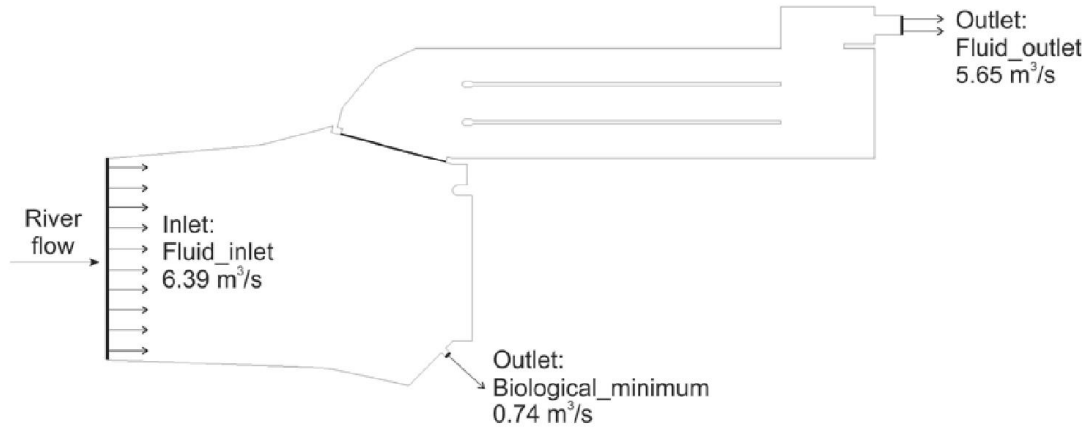


Figure 2: Boundary conditions

Where  $U$  is the average value of the velocity,  $u$  is fluctuating value of the velocity,  $P$  is pressure,  $\rho$  is density, and  $\delta_{ij}$  is Kronecker delta, with a value of 1 if  $i=j$  or 0 in other cases. A Reynolds stress term from the equation (1) is modeled by approximating Boussinesq':

$$-\rho u_i u_j = \rho v_T \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (2)$$

Where the variable  $k$  is turbulent kinetic energy, and  $v_T$  is turbulent eddy viscosity. By inserting equation (2) into the equation (1) is obtained the final form of equation:

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left[ - \left( P + \frac{2}{3} k \right) \delta_{ij} + v_T \frac{\partial U_i}{\partial x_j} + v_T \frac{\partial U_j}{\partial x_i} \right] \quad (3)$$

The first member of the equation on the left of the previous equation is a non-stationary member, which is set in the model is zero, because the velocity of the water in the water intake is defined as stationary. For calculation of the turbulent eddy viscosity, in equation (2) the  $k$ - $\epsilon$  model from [11] was used:

$$v_T = c_\mu \frac{k}{\epsilon^2} \quad (4)$$

Where the turbulent kinetic energy  $k$  modelled as:

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{v_T}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + P_k - \epsilon \quad (5)$$

Where:

$$P_k = v_T \frac{\partial U_j}{\partial x_i} \left( \frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) \quad (6)$$

Dissipation of turbulent kinetic energy  $k$  is denoted by  $\epsilon$ , and modeled as:

$$\frac{\partial \epsilon}{\partial t} + U_j \frac{\partial \epsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{v_T}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + C_{\epsilon 1} \frac{\epsilon}{k} P_k + C_{\epsilon 2} \frac{\epsilon^2}{k} \quad (7)$$

Where constants in the  $k$ - $\epsilon$  model have the following standard values:  $c_\mu = 0.09$ ;  $C_{\epsilon 1} = 1.44$ ;  $C_{\epsilon 2} = 1.92$ ;  $\sigma_k = 1.0$ ;  $\sigma_\epsilon = 1.3$ ;

Semi - Implicit Method for Pressure - Linked (SIMPLE) is used for solving described a model of the water intake.

## 4. RESULTS

For simulating the results of the side water intake in the paper, the commercial computational fluid dynamics (CFD) software package Ansys is used. This software package is able to use two modules to simulate fluid flow: Ansys Fluent and Ansys CFX. In the case of 3D flow, it is possible to use both modules, CFX and Fluent. When the 2D flow model is set, as is the case in this study, it is possible to use only module Fluent.

Ansys Fluent, like other CFD software packages, solves the fundamental equations of fluid flow [12], conservation of mass and momentum, known as Reynolds-averaged Navier-Stokes (RANS) equations. The Software solves additional equations that represent the turbulent characteristics ( $k$ - $\epsilon$ ) to determine eddy viscosity and mixing coefficients. Ansys Fluent discretized partial differential equations using finite volume method, and solve the resulting algebraic equations by implicit methods [13], [14], and [15].

### 4.1. Existing state

The result of the simulation of the current state of the water intake of small hydropower plant is shown in Figure 3. Shown is a planar velocity field of the flow of water in the range from 0 to 1.35 m/s. It may be noted that the water velocity in the chambers of the settling basin drastically different. Top speed, i.e. the flow is realized in the third chamber of settling basin. In the secondary chamber is achieved somewhat lower speed, while in the first chamber, the water is almost not moving, and the flow rate is approximately zero, except for a few local fields of velocity different than zero (see Figure 3).

### 4.2. Proposed technical solutions

To overcome the problem of uneven flow in the three chambers of the settling basin, 12 technical solutions are proposed and shown in Figures from 4 to 15. The Settlement provides for a change in a part of settling basin between the lattice and the chambers of the settling basin. The proposed technical solutions are characterized by simplicity, the minimum price and the lowest possible pressure drop. The aim of the presented results that are based on established criteria is to determine the best technical solution to reduce the difference in speed and flow in the chambers of the settling basin.



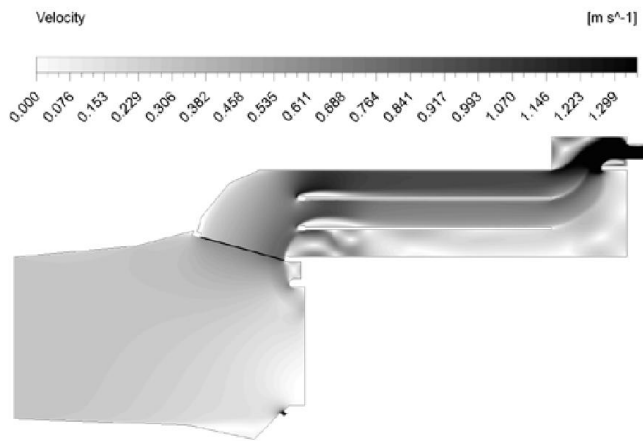


Figure 3: Existing state

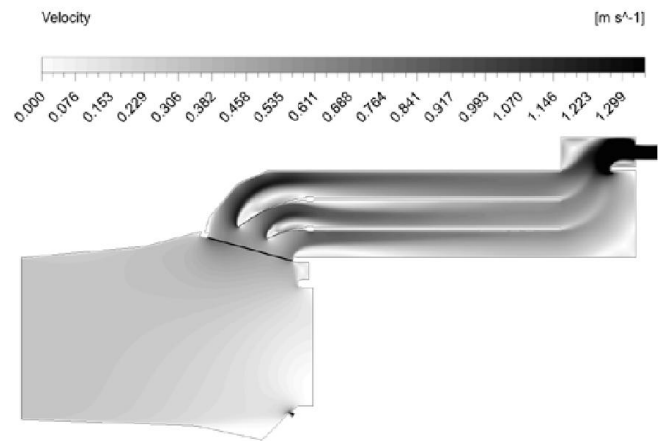


Figure 7: Solution 4

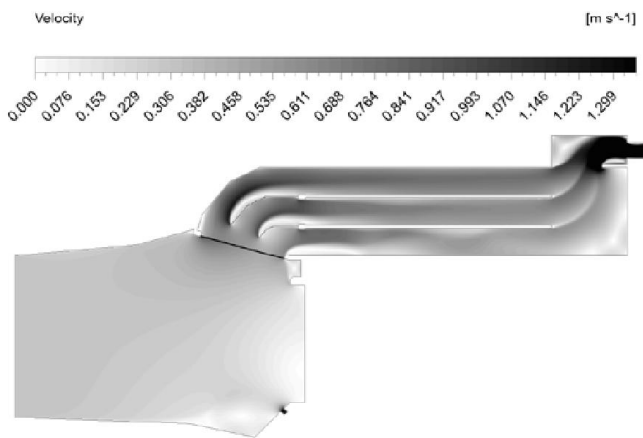


Figure 4: Solution 1

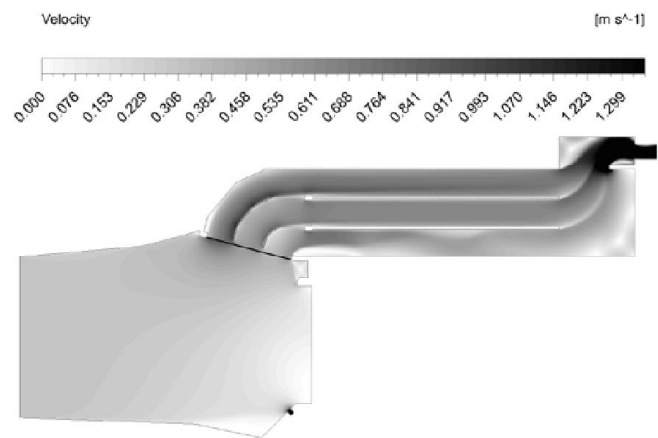


Figure 8: Solution 5

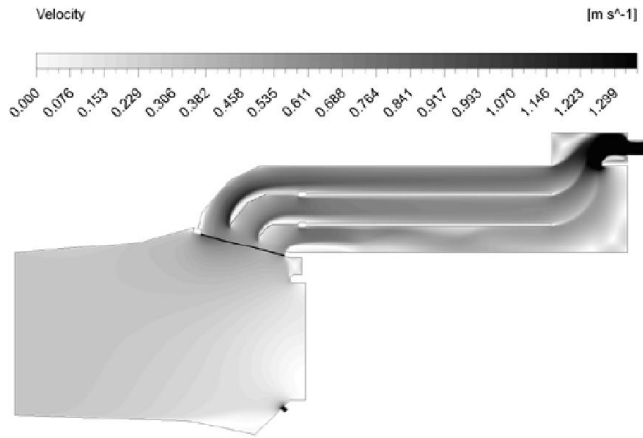


Figure 5: Solution 2

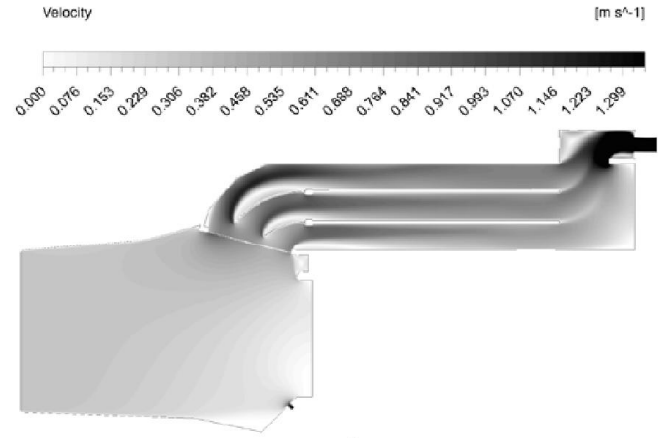


Figure 9: Solution 6

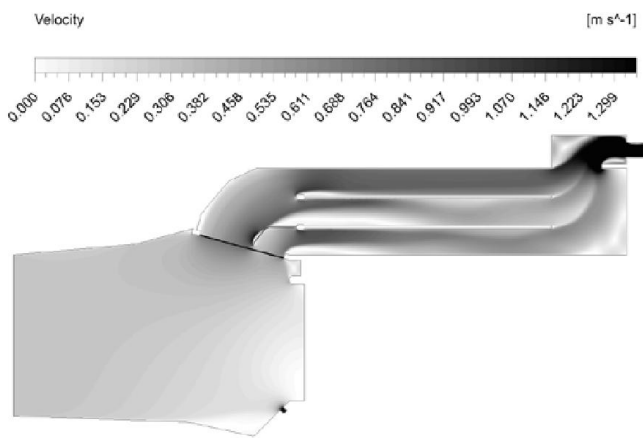


Figure 6: Solution 3

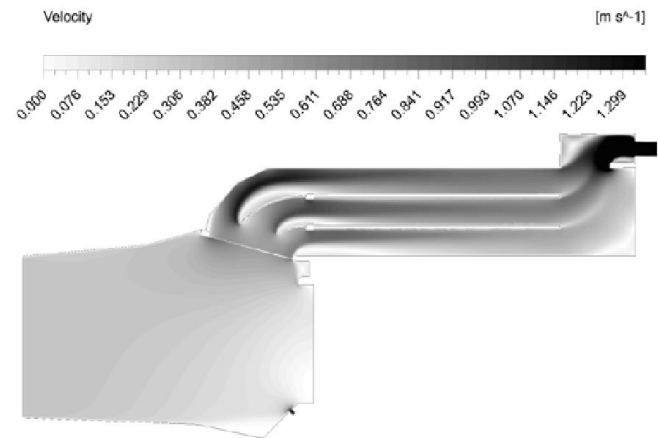


Figure 10: Solution 7

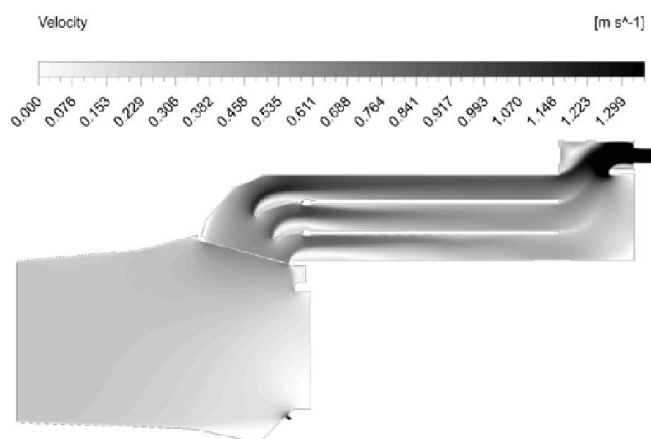


Figure 11: Solution 8

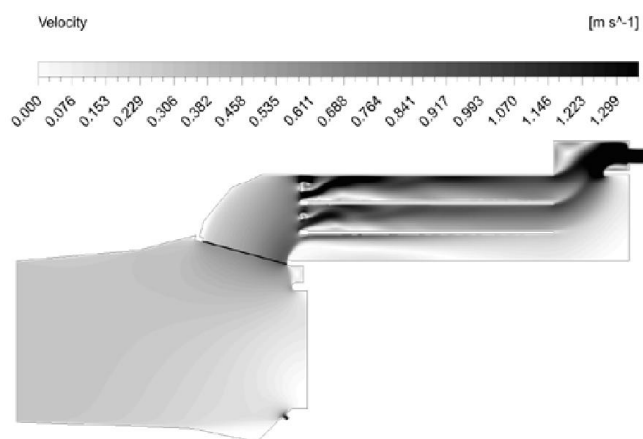


Figure 15: Solution 12

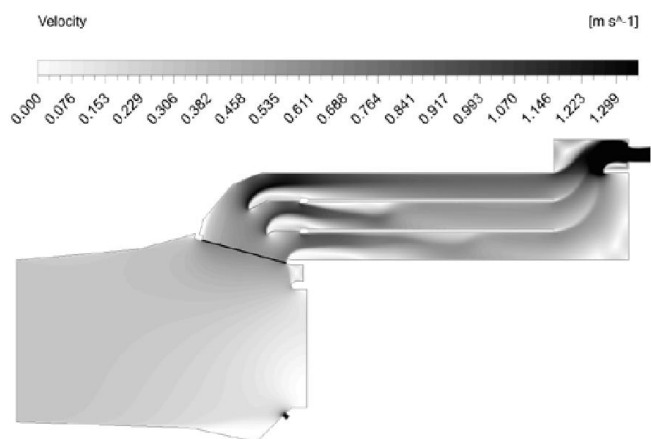


Figure 12: Solution 9

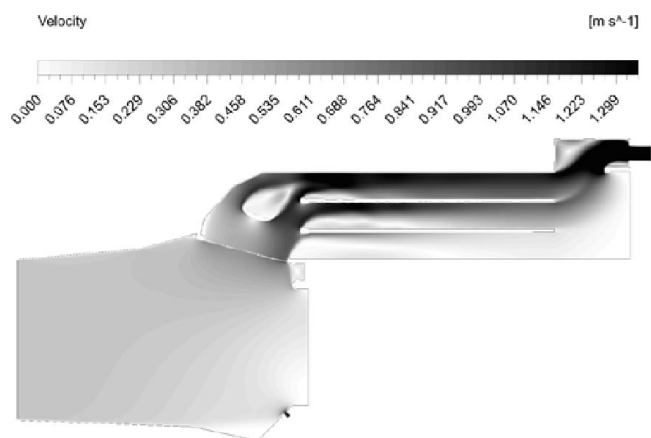


Figure 13: Solution 10

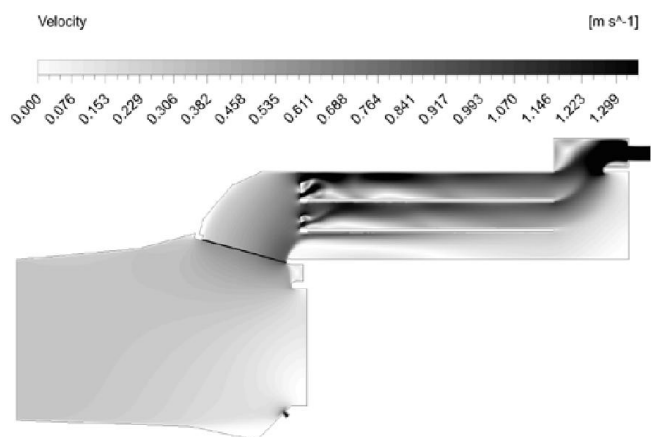


Figure 14: Solution 11

If one looks at the current state of the side water intake (Figure 3) it can be seen that the velocity in the area of the lattice approximately evenly distributed. The problem arises because of the intensity of the speed that is on average between 0.611 and 0.688 m/s at the entrance. Based on these facts were proposed technical solutions for adding a router and a barrier in the area between the lattice and the chambers of the settling basin. The idea is to determine the optimal shape and length of the router of water in order to achieve the desired effect, equalizing the water speed in the chambers of the settling basin.

In the figures 13, 14, and 15 are shown the results of simulation of the proposed technical solutions with the increase of the resistance in the first and second chamber (see Figures 14 and 15), wherein the partition wall is greater in the first chamber in the relative to the wall in the second chamber, for the reason that in the analysis of the current state (see Figure 3.) first chamber has the highest speed. The difference between solutions 11 and 12 is in the position of placing a barrier in relation to the chambers of the settling basin. The solution 10 (in Figure 13.) represent the reconstruction in order to create a resistance to the central part of the settling basin, between the lattice and the chambers, where is the main direction of flow of the water towards the first chamber, and a smaller portion to the second chamber. This three solutions would cause a deterioration compared to the current situation, so it are rejected.

The solution 3 (Figure 6) provides for the installation of a router between the second and the third chamber. This solution increases the velocity in the third chamber, but significantly reduced in the second chamber. The Solution 4 (Figure 7.) provides for the installation of special shape router, but this solution increases the speed in the first chamber.

Solutions 1, 2, 5, and 9 are shown in Figures 4, 5, 8, and 12, respectively. This group of solutions represents the installation of the flow router with the criterion that the width of new chambers be equal. The differences between these four solutions are in lengths of dividers. Here can be seen that for a solution is beneficial that the routers as long as possible. Solutions 1, 2, and 5 provide approximately the same result, because the differences between the lengths of the flow deflector are very small. These three solutions, which essentially boils down to one solution with different router lengths, represent a potential solution to the observed problem.

The remaining solutions 6, 7, and 8, are presented in Figures 9, 10, and 11. These three solutions provide for the installation of the router in the form of an arc, wherein the different lengths between the three solutions. These three solutions have led to substantial improvements compared to the current situation, but speeds continue to vary, so that the speed is higher in the first chamber, and in a third smaller, relative to the rate flow of the water in the second chamber of settling basin, wherein is exception is the solution 6 (Figure 9) where the speed in the chambers quite balanced.

Based on the analysis of all the proposed solutions, the best results in the equalization of speed in the chambers of settling basin can be achieved by installing a router as provided by solutions 1 (Figure 4.) and 6 (Figure 9.).

## 5. CONCLUSION

The small hydropower plant with installed capacity of 1.25 MW, which is in operation since the end of 2013 was observed in this study. This hydropower plant is run of the river type with lateral water intake. In the exploitation conditions, it has been observed that the flow of water in the three chambers of the side intake settling basin differs drastically, so that a single chamber is almost no flow.

The aim of this study has been to propose a technical solution for the reconstruction of water intake so as to equalize the speed and flow in the chambers of the settling basin. Criteria for selection of the optimal solution are: minimum price, simplicity, and the lowest possible pressure drop. To solve this problem a two-dimensional CFD model was formed, that is generated by a grid with 217979 elements and 227339 nodes, and by defined boundary conditions.

In order to solve the defined model is used the Ansys Fluent software package, on the basis of equations of a turbulent flow of the fluid (RANS) and additional equations which describe the turbulent characteristic ( $k-\epsilon$ ). The Semi-Implicit Method for Pressure-Linked (SIMPLE) algorithm was used for solving the model in a Fluent module of Ansys software package. As a result, showing the 12 simulations of the proposed technical solutions. Based on the results, several technical solutions lead to a reduction in the difference in the speeds of which are the best results achieved with solutions 1 and 6.

Continued research should consist of drafting and 3D simulation model of the side water intake, and the inclusion of sediments in the fluid flow.

## ACKNOWLEDGEMENTS

This work was conducted within the project TR 33027 supported by the Ministry of Education, Science and Technological Development of the Republic of Serbia.

## REFERENCES

- [1] Pandey, Bikash, and Ajoy Karki. *Hydroelectric Energy: Renewable Energy and the Environment*. CRC Press, 2016.
- [2] REN21, Renewables 2016 Global Status Report. REN21 Secretariat, Paris, 2016.
- [3] Jovančić, P. D., Kolonja, B., Ignjatović, D., Tanasijević, M., Madžarević, A., & Krstić, V. (2016). Energy resources in the Republic of Serbia: Development policy. *Energy Sources, Part B: Economics, Planning, and Policy*, 11(11), 1020-1026.
- [4] <http://www.srbijavode.rs/sr-latin/home/Aktuelno/mhe.html> (in Serbian)
- [5] Pekez, J., Radovanovc, L., Desnica, E., & Lambic, M. (2016). The increase of exploitability of renewable energy sources. *Energy Sources, Part B: Economics, Planning, and Policy*, 11(1), 51-57.
- [6] <https://energija.wordpress.com/2009/01/05/katastar-malih-hidro-elektrana/> (in Serbian)
- [7] <http://www.elektrosrbija.rs/me/images/dokumenti/Katastar%20MHE%20u%20Srbiji.pdf> (in Serbian)
- [8] Lai, Y. G., Weber, L. J., & Patel, V. C. (2003). Nonhydrostatic three-dimensional model for hydraulic flow simulation. I: Formulation and verification. *Journal of Hydraulic Engineering*, 129(3), 196-205.
- [9] Khan, L. A., Wicklein, E. A., Rashid, M., Ebner, L. L., & Richards, N. A. (2004). Computational fluid dynamics modeling of turbine intake hydraulics at a hydropower plant. *Journal of Hydraulic Research*, 42(1), 61-69.
- [10] Thompson, Joe F., B. K. Soni, and N. P. Weatherill. "Handbook of Grid Generation CRC Press." Boca Raton, Florida (1999).
- [11] Olsen, Nils Reidar B. "Hydroinformatics, fluvial hydraulics and limnology." *Norwegian University of Science and Technology. Trondheim, Noruega* (2004).
- [12] Khan, Liaqat A., Edward A. Wicklein, and Mizan Rashid. "A 3D CFD model analysis of the hydraulics of an outfall structure at a power plant." *Journal of Hydroinformatics* 7.4 (2005): 283-290.
- [13] Olsen, Nils Reidar B. "Computational fluid dynamics in hydraulic and sedimentation engineering." *The Norwegian University of Science and Technology, Trondheim* (1999).
- [14] Rodi, W. "Turbulence Models and Their Applications in Hydraulics, International Associate for Hydraulic Research, Delft, The Netherlands." (1980).
- [15] Lomax, Harvard, Thomas H. Pulliam, and David W. Zingg. *Fundamentals of computational fluid dynamics*. Springer Science & Business Media, 2013.