Available online at www.CivileJournal.org

# Civil Engineering Journal

Vol. 3, No. 11, November, 2017

# Investigating Effect of Different Parameters of the Submerged Vanes on the Lateral Intake Discharge Located in the 180 Degree Bend Using the Numerical Model

A. Sarhadi [a*], E. Jabbari [b]

[a] *Masters Student of Civil Engineering, University of Qom, Iran.*

[b] *Assistant Prof., Dept. of Civil Engineering, University of Qom, Iran.*

Received 10 October 2017; Accepted 21 November 2017

**Abstract**

Intakes are widely used for flow diversion and its control in the open channels or rivers. During passing flow, part of the suspended sediment along with the flow enters the lateral channel and deposits in the lateral intake channel entrance, causing a change in the direction of the flow line towards the shore in front of the reservoir, which reduces the intake efficiency. Submerged vanes are small hydraulic structures that, by creating a secondary flow in their downstream, cause changes in the flow pattern and guide line to the drainage span, and the most important parameters affecting sediment input to the waterfall is the ratio of flow rate. Investigating a laboratory model has high costs and times, which in some cases cannot be justified, therefore, suitable numerical models can be proposed for such options. In this study, using Flow3D, three-dimensional numerical modeling of the flow was calibrated and verified using existing data and numerical modeling accuracy, the relative error of the numerical model was determined. In this study, all effective parameters including submerged vanes type, submerged vanes number, submerged vanes size and Froude number changes in the main channel and type of submerged vanes layout have been investigated. The results of the numerical model show that the angle of inclination of 60 degrees in the entrance intake and the chassis layout in the Froude numbers 0.21-0.33 will result in the most lateral intake discharge.

*Keywords:* Submerged Vanes; Lateral Intake; Discharge; Bend; Flow3D.

## 1. Introduction

At the juncture of canals, the two main and sub-main flows are diverted from their original path and the flow pattern and properties are changed. Change in the water surface profile and flow depth, velocity distribution, stagnation zone, channel contraction, energy loss and formation of hydraulic jump are among the most important hydraulic variables at this point. At the stagnation point which is created at the upstream corner of the juncture, the first contact between the two main and sub-main flows occurs and the flow velocity is nearly zero at this zone. The diversion zone, is where the main canal flow is deviated from its main path and approaches the opposite juncture wall. The sub-main canal flow distances from the inner canal wall after joining it and creates a zone named as the separation zone, where the flow velocity is very low and sedimentation occurs there. At the maximum velocity zone or the contraction zone, the flow velocity is highly increased and where the shear stress exceeds the critical one local erosion occurs there. Estimation of the local erosion at the juncture of the rivers and canals is among the most important problems in hydraulic engineering.

Use of the submerged vanes technique which are the developed and modified form of the deep panels was investigated for the first time by Odgaard and Kennedy (1983) at the hydraulic institute of Iowa University for Prevention of the outer

* Corresponding author: ali.sarhaddy@gmail.com

http://dx.doi.org/10.28991/cej-030947

bank erosion in rivers [1]. Later, Nakao et al. (1990) were the first to incorporate submerged vanes for sediment control at the intake of the pumping station of a power plant located on Missouri River using the laboratory model [2]. In the studies performed by Wang and Odgaard (1993) [3], Johnson et al. (2001) [4], Barkdoll 2003 [5], Johnson et al. (2003) [6], Michell et al. (2006) [7] entrance of sediments to the lateral intakes is comprehensively studied.

To investigate the local scouring at the juncture of open channels, use has been made of the laboratory studies by Borghei and Jabbari Sahebari (2010) [8]. In the studies by Sharma et al. (2015), using the basic laboratory studies and numerical analysis tools (CFD), investigation and optimization of the flow pattern in implementing submerged vanes in the canal is performed [9]. Ouyang and Lu (2016) studies include investigation and optimization of the spacing between the vanes and the angle of the vanes to the flow direction [10]. Dey et al. (2017), implementing the ADV velocimeter at the 180 degrees bend investigated changes in the bed profile with respect to the changes in velocity using the submerged vanes [11].

Reviewing the previous studies it is found that incorporation of the submerged vanes at 180 degrees bends and at the juncture of lateral intake for investigation of their effect on the diverted discharge has not been comprehensively studied. Therefore in this study by changing the various influential parameters in the submerged vanes attempt is made to determine the diverted discharge to the lateral intake using the FLOW3D numerical model.).

## 2. Materials and Methods

Numerical simulation of flow in canals and their basins in the design process is of great importance. The experience of researchers concerning this type of simulations has shown that among the available software packages, the FLOW3D software has a better capability in modelling of this type of hydraulic conditions. In this study to investigate the effects of various parameters of the submerged vanes on the diverted discharge of the lateral intake in a 180 degrees bend, the numerical modelling software FLOW3D (ver. 10.1) has been incorporated. Calibration and validation of the simulation results in this software are done based on the laboratory studies. Geometry of the laboratory models is made by creation of the identical numerical analysis conditions which are similar to the hydraulic model, and the flow field hydraulic parameters are investigated and compared in these conditions. What that is simulated in this research corresponds to the laboratory model incorporated in the Barmaki et al. studies which were performed in 2013 on a curved flume of 180 degrees and with a rectangular section of 0.6 $m$ width, 0.4 $m$ height and a relative curvature ratio; $R_c/B$ equal to 4.70, where the central radius of the curvature was equal to 2.8 $m$. The invert level of the main canal and that of the diverted canal were equal and the needed water was supplied by an underground water tank utilizing a 6inch pump with maximum capacity of 50 $Lit/s$ which discharged water into the flume [12]. A view of this laboratory model is shown in the Figure 1.

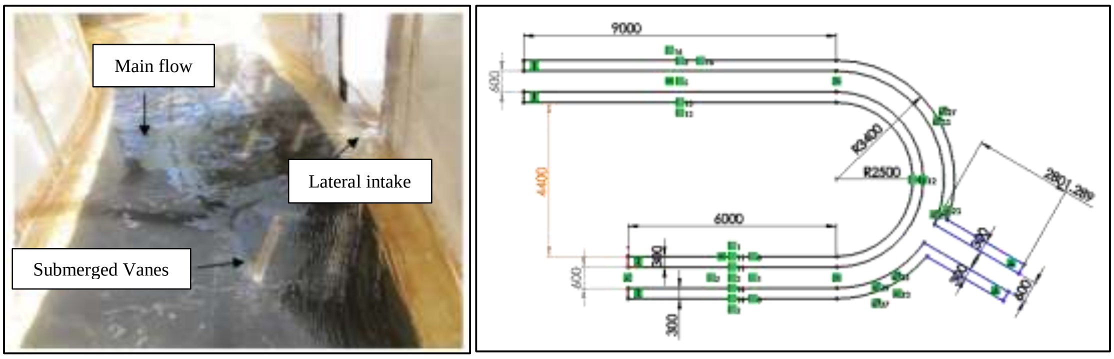


**Figure 1. Geometric properties and a view of the 180 degrees physical model and the lateral intake**

They performed the experiments by installing 4 intakes with angles of 45, 60, 75 and 90 degrees at 4 positions of 30, 70, 100 and 120 degrees on a curved canal with 4 Froude number values of 0.21, 0.26, 0.31 and 0.37. In all the experiments two rows of the submerged vanes were installed at the entrance of the intake with constant dimensions of 10.8 cm length and 3.6 cm height at a constant angle of 20 degrees.

## 3. Governing Equations of Fluid Dynamics and Flow3D Model

FLOW3D is an appropriate model for complex fluids problems. This numerical model is widely used, particularly for unsteady 3-dimensional flows with free level and complex geometry. In this model, finite volume method is used in regular rectangular grid generation. Due to using finite volume method in a regular grid, the form of the employed discrete equations is similar to discrete equations in finite difference method. Accordingly, FLOW3D enjoys first and second-order reliability methods which are explained in the following. Also, this software uses five turbulence models such as k-ε and RNG. In FLOW3D, two methods have been simultaneously used for geometrical simulation. The first method is volume of fluid (VOF) which is used to show the behaviour of fluid at free level. The second method is fractional area-volume obstacle representation (FAVOR) which is used to simulate solid levels and volumes such as geometrical boundaries.

Equations governing fluid flow are obtained from the law of conservation of mass and the law of conservation of momentum. These equations are in the form of partial differential equations. In general, to obtain flow equations, three steps should be considered: selecting accurate base laws, applying laws by an appropriate model and adopting mathematical equations showing the mention physical laws. The main equations to simulate 3-dimensional flow are three differential equations including continuity relations and movement size in x; y and z directions. A flow continuity equation is obtained from the law of conservation of mass and writing balance equation for a fluid element. General continuity equation is presented as Equation 1.

$$V_f\frac{\partial\rho}{\partial t}+\frac{\partial}{\partial x}(\rho uA_x)+\frac{\partial}{\partial y}(\rho vA_y)+\frac{\partial}{\partial z}(\rho wA_z)=0 \tag{1}$$

Where $V_f$ indicates the fraction of open volume to flow; ρ indicates fluid density; (u, v, w) indicate velocity components in the directions of (x, y, z); Ax indicates the fraction of open level in x direction; Ay and Az indicate the fraction of open level in y and z directions.

Fluid movement equations with velocity components of (u, v, w) in three different directions, i.e. Navier-Stokes equations are presented as following:

$$\begin{aligned}
\frac{\partial u}{\partial t}+\frac{1}{V_F}\left(uA_x\frac{\partial u}{\partial x}+vA_y\frac{\partial u}{\partial y}+wA_z\frac{\partial u}{\partial z}\right)&=-\frac{1}{\rho}\frac{\partial p}{\partial x}+G_x+f_x\\
\frac{\partial v}{\partial t}+\frac{1}{V_F}\left(uA_x\frac{\partial v}{\partial x}+vA_y\frac{\partial v}{\partial y}+wA_z\frac{\partial v}{\partial z}\right)&=-\frac{1}{\rho}\frac{\partial p}{\partial y}+G_y+f_y\\
\frac{\partial w}{\partial t}+\frac{1}{V_F}\left(uA_x\frac{\partial w}{\partial x}+vA_y\frac{\partial w}{\partial y}+wA_z\frac{\partial w}{\partial z}\right)&=-\frac{1}{\rho}\frac{\partial p}{\partial z}+G_z+f_z
\end{aligned} \tag{2}$$

In these equations, $(Gx, Gy, Gz)$ indicates mass acceleration and $(fx, fy, fz)$ indicate viscosity accelerations.

## 4. Numerical Modeling

For simulation of the flow in this flume, the gravitational constant value was introduced as $g = -9.81\ m^2/s$ to the software. To account for the effect of jet energy dissipation by the air entrance, the air entrance model with a density of $1.2\ kg/m^3$ and a shear stress coefficient of $0.073$ was selected (Yamini et al. 2015) [13]. Flow in the canal exhibits a relatively turbulent behaviour and therefore in all the simulations in this research the flow field is solved using the $RNG$ turbulent model according to the FLOW3D recommendation. The reason for recommending this turbulence model is due to its features and advantages over such models as $K-\varepsilon$ model. Rostami also conducted a research on the turbulence models using the FLOW3D software and concluded that the $RNG$ turbulence model yields more precise result with respect to other turbulence models. Although in this research to ensure the validity of the existing difference between the corresponding results of the two turbulence models ($RNG$ and $K-\varepsilon$), simulations were performed using the two models of $K-\varepsilon$ and $RNG$ after calibrating the entire model. it was observed that there is not a significant difference between the corresponding results of the two models but the K-ε model was much more time consuming than the $RNG$ model. For the flow simulation in the canal, two mesh blocks were defined. In the first block for the boundary condition of the input flow, a volume of flow rate equal to $50\ L/s$ and for the banks and the bottom the Wall boundary condition were selected. Also for the upper face and the outflow the symmetry condition were utilized. In the second block the symmetry condition was used for the input face and the outflow boundary, and for the banks and the bottom the Wall condition was utilized. In the Figure 2. a view of these conditions is depicted. The fluid used in this research is water in the laboratory temperature (about 15 Celsius degrees).

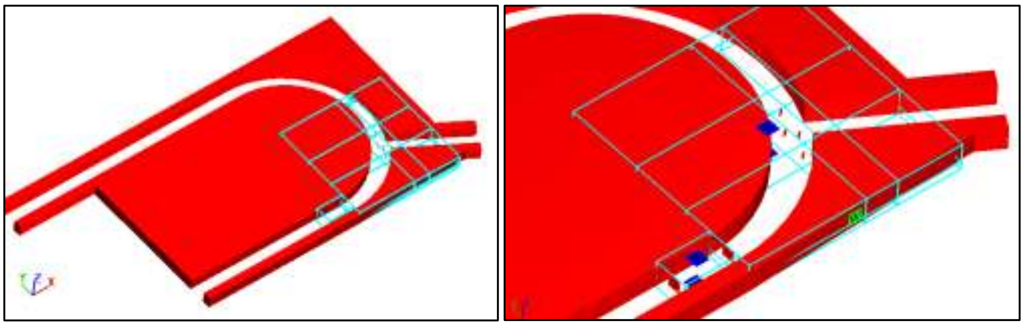

**Figure 2. Meshing blocks and applying the boundary conditions in the FLOW3D numerical model**

FLOW3D has a different approach toward the solid boundaries and meshing in comparison to other CFD packages. To specify the boundaries of the meshing, a number of blocks were specified where all the canal dimensions and the open space within it were defined and all the details were given in those blocks. In the Figure 3. the numerical model and the FAVOR method for meshing are demonstrated. Perhaps creation of proper solid boundaries in the meshing of the numerical model is the greatest limitation and its appropriate selection could enhance the computations' precision. In this study various meshing schemes were examined and the best one was selected for the study.

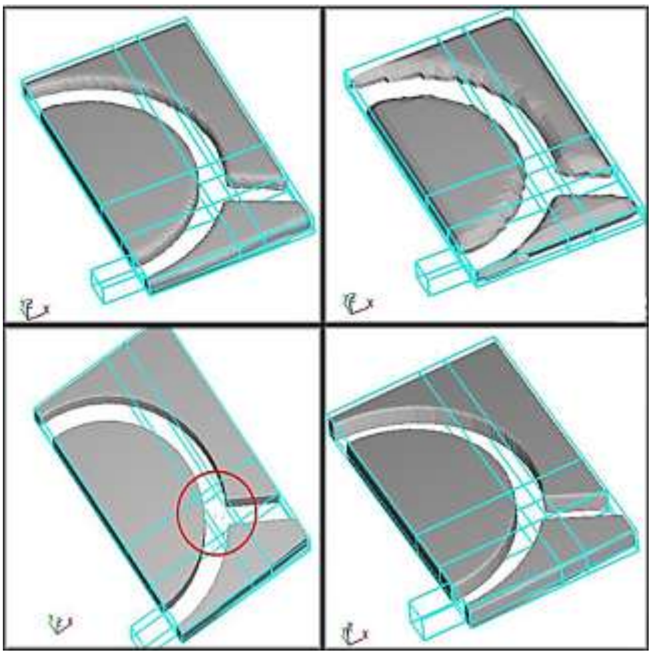

**Figure 3.Performance of the FAVOR method with solid boundaries and final selected mesh.**

As seen in Table 1, the cell dimensions are getting smaller in each test. It should be noted that the mention tests did not result in an optimal and appropriate state just by changing the cell dimensions. But in addition to the cell dimensions other factors including the simulation time, boundary conditions of the first block, use of mesh plans in sensitive areas like the submerged vane area, and change in the initial conditions had been very effective which were separately considered. Finally, with respect to different results obtained from various tests, the best test model which is test no. 5 in Table 1. was selected.

**Table 1. Meshing characteristics of the various cells used in simulation of the flow in the flume**

| # Test | X Direction $(cm)$ | Y Direction $(cm)$ | Z Direction $(cm)$ | Total Mesh |
|---|---|---|---|---|
| 1 | 5 | 6 | 5 | **854423** |
| 2 | 4 | 5 | 4 | **723322** |
| 3 | 2 | 3 | 3 | **1214163** |
| 4 | 5 | 3 | 2 | **1146726** |
| 5 | 2 | 2 | 2 | **1475236** |
| 6 | 1 | 2 | 2 | **2987512** |

## 5. Results and Discussion

In the present research, the relative diverted discharge was simulated by changing the Froude number and consequently various flow depths. Therefore in this section, the corresponding results of the flow depth in the numerical model with respect to change in the Froude number (0.21, 0.26, 0.31 and 0.37) at different installation angles of submerged vanes ranging from 10 to 40 degrees are presented. Also the results are compared to those of the laboratory results. In the Figure 4, the stages of flow formation in the curved canal in the case of 20 degrees angle of the submerged vanes and Froude number 0.21 are presented.

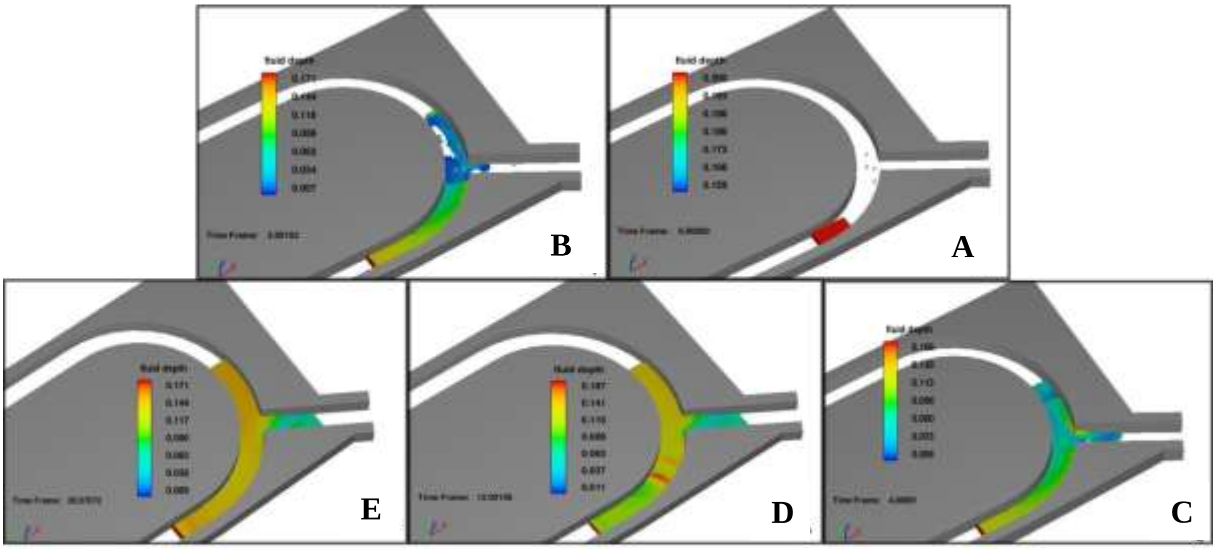


**Figure 4. Flow depth value in the numerical model at various time steps (A to E) in the curved canal using submerged vanes with 20 degrees angle.**

Now we could calculate the simulation discharge value With respect to the flow depth, and then compare the relative diverted flow value with that of the laboratory model at the entrance of the lateral intake. The diverted discharge value is calculated from the equation 3:

$$Q_r = \frac{Q_D}{Q_m} \quad (3)$$

In this expression $Q_m$ is the total discharge which is equal to the diverted and non-diverted discharges in the canal and $Q_D$ is equal to the diverted discharge in the lateral canal. In the FLOW3D numerical model there are two different turbulence models; the standard $K - \varepsilon$ and $RNG$ for the free surface flow. In simulation of the flow using various CFD models, selection of a turbulence model which is appropriate for the flow condition is one of the most important parts of the computational fluid dynamics. Thus these two turbulence models are incorporated and compared to each other for the flow simulation in the lateral intake canal. In the Table 2. the results corresponding to the flow simulation are compared and assessed using the standard $K - \varepsilon$ and RNG turbulence models.

**Table 2. Consumed time and simulation error in the lateral intake canal for various turbulence models of $RNG$ and $k - \varepsilon$**

| $Fr = 0.37$ | $Fr = 0.31$ | $Fr = 0.26$ | $Fr = 0.21$ | | Froude Number |
|---|---|---|---|---|---|
| 42223 | 42223 | 42223 | 42223 | K-ε | **Simulation times ($S$)** |
| 32478 | 32478 | 32478 | 32478 | RNG | |
| 47.8 | 50.7 | 62.8 | 73.7 | K-ε | **Lateral Intake Discharge ($Q_r$)** |
| 48.3 | 51.2 | 63.8 | 74.5 | RNG | |
| 50 | 55 | 68 | 88 | EXP | |
| 4.4 | 7.81 | 7.64 | 10.12 | K-ε | **Relative error rate (%)** |
| 3.4 | 6.17 | 6.9 | 9.14 | RNG | |

Considering the Table 2, it is observed that the difference between two turbulence models of RNG and K-ε is not significant and the corresponding results of these two models for the flow simulation in the lateral intake canal are very close to each other. The results of the numerical model show that the mean error for all the Froude numbers in the K-ε turbulence model is about 1.1% higher than that of the RNG turbulence model. Therefor the consumed time for the flow simulation in the lateral intake canal could be a good criterion in selecting the appropriate turbulence model in this part. The required time for the turbulence model of K-ε is about 1.3 times that of the RNG turbulence model, while there is a negligible difference between their relative results. On the other hand the reason for using this turbulence model is due to the features and advantages of this model over against such models as $K - \varepsilon$. This model is improved due to having an additional term in ε equation for analysis of rapidly strained flows and also flows over surfaces with high geometrical variations. Also this model has a great capability in simulation of the transient flows. According to the comparison made by Lan et al. (2007) between various turbulence models using the FLOW3D software, the RNG turbulence model had more accurate results with respect to those of other turbulence models concerning the lateral intake canal. Therefor the

RNG turbulence model could be incorporated with enough confidence. For a more precise investigation of the relative discharge results corresponding to the lateral intake canal, the diagram corresponding to the discharge changes for various Froude number values and considering the two cases of laboratory and numerical studies has been presented in the Figure 5. It should be noted that for computing the relative error values the equation 4, is used:

$$Error\ \% = 100 * \frac{X_P - X_M}{X_P} \tag{4}$$

In the Equation 4, $X_P$ denotes the real values and $X_M$ denotes the simulated values of the intended parameter.

Presence of the intake causes flow division at the intake upstream of the canal, where a portion of the flow enters the intake and the other portion continues its path in the main canal. Calculation of the mean relative error of the numerical model for the lateral intake discharge shows that the mean relative error for all the Froude number values is about 6.4%. This amount of error is acceptable considering the calibration stages, and other numerical models would be run using these boundary conditions and the turbulence model conditions.

In this section in order to compare the relative diverted discharge corresponding to the case of submerged vanes at different installation angles with the case of without submerged vanes, attempt is made to perform a new simulation by removing the submerged vanes.

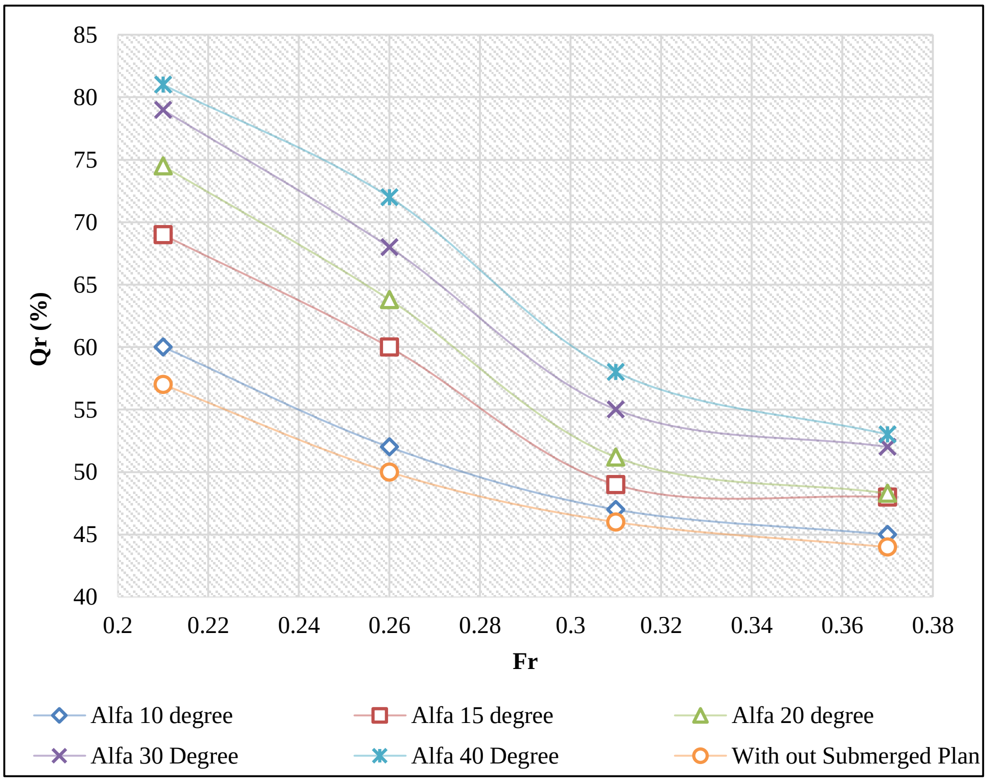


**Figure 5. Comparison of the relative diverted discharge in the case of with and without submerged vanes and the laboratory model**

Considering the diagram in Figure 5. it is found that the relative diverted discharge in the case of without submerged vanes and for all Froude numbers, is smaller with respect to other cases. In other words the flow in the lateral intake canal is smaller when there are no submerged vanes. Reduction of the flow in the lateral intake canal is intensified at higher Froude numbers, as in this case the flow velocity is higher in the main canal and the possibility of flow in the lateral canal is decreased.

It should be noted that in cases where the flow has suspended particles and sediments, the inflow of sediments in the case of submerged vanes depends on the dimensionless parameters such as the Froude number, angle of intake and ratio of discharge. Here the role of submerged vanes should be re-examined and decision be made for the critical conditions of entering minimum amount of the sediments into the lateral canal or entrance of the maximum discharge.

In the second step, the positioning angle of the submerged vanes in the case of more than 40 degrees till reaching the maximum discharge in the numerical model is considered. So by changing the angle of positioning for the cases of 50, 60, 70 and 80 degrees the variation of discharge was modeled. The diagram of discharge variation based on these models is given in Figure 6.

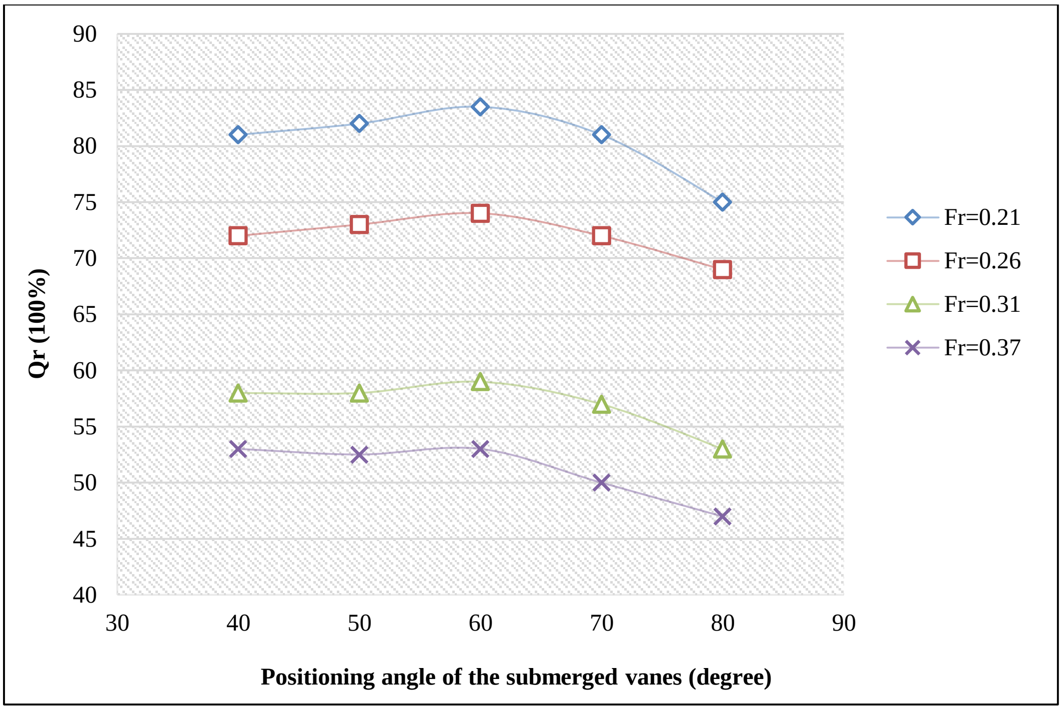


**Figure 6. Variation of the relative diverted discharge in the lateral canal with respect to further increase of the positioning angle of the submerged vanes in the main canal**

As seen in Figure7, further increase in the positioning angle of the submerged vanes in the case of greater than 40 degrees causes changes in the lateral intake discharge, so that due to increase of the angle from 40 to 50 degrees the mean relative diverted discharge for all the Froude numbers changes from 66% to 66.375% which shows a very small increase in such condition. Also by increase in the angle from 40 to 60 degrees the mean discharge shows an increase from 66% to 67.375%. Thus as seen in the Figure 6. the best positioning angle to achieve the maximum diverted discharge is 60 degrees which occurs for all the Froude numbers. It should be noted that for other Froude numbers we have the reducing trend of the diverted discharge. The reason for the reduced inflow to the diverted canal lies in the flow blockage at the entrance to the lateral canal. By increasing the positioning angle of the submerged vanes to 70 and especially 80 degrees no diversion takes place towards the lateral intake and the submerged vanes act as blockers of the flow in the main canal. This flow blockage causes reduced velocity and consequently reduced values of the Froude number after passage of flow from the submerged vanes. By reduction in the Froude number value, certainly according to the Figure 6. the diverted discharge to the lateral canal would decrease, too. In the diagram of the following figure the general trend of changes in the relative discharge per all changes in the modelled positioning angles is depicted.

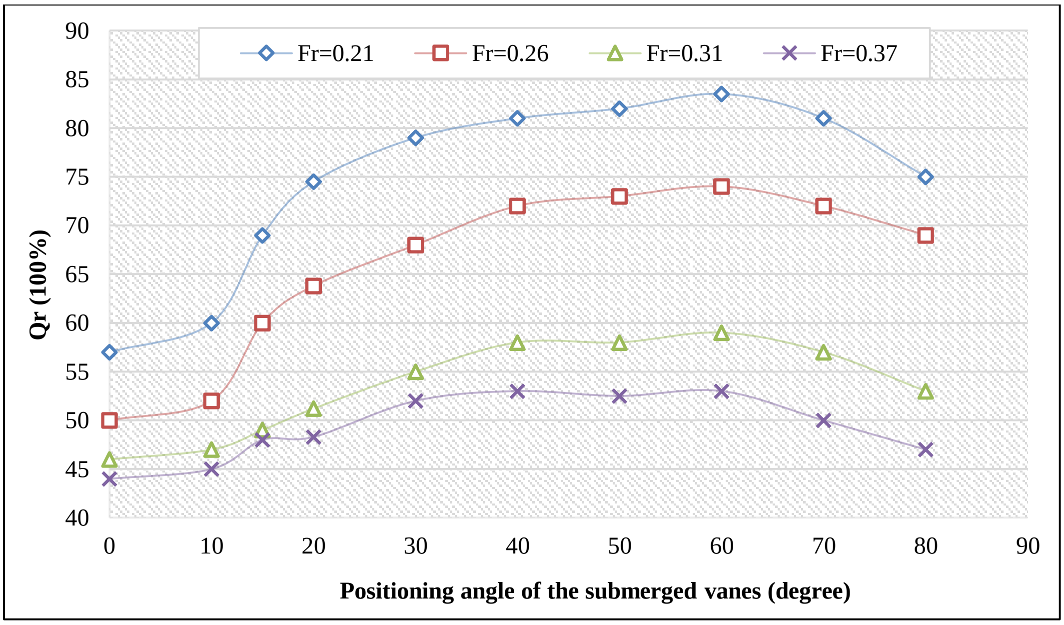


**Figure 7. Determining changes in the relative diverted discharge in the lateral canal with respect to changes in the positioning angle of the submerged vane in the main canal**

As was presented in the previous sections, the positioning angle of the submerged vanes could affect the amount of diverted discharge especially for smaller Froude number values these changes are more visible. Based on the numerical model results, the maximum discharge value in the lateral intake canal corresponds to the submerged vanes positioning angle of about 60 degrees. For this reason to investigate the effect of arrangement of the vanes and their number, the geometric characteristics of 60 degrees angle of positioning are selected and first in this special arrangement the number of vanes is increased from 4 to 6 and then to 8 in this arrangement. In the Figure 8. the 6 and 8 submerged vanes are shown for the positioning angle of 60 degrees, respectively.

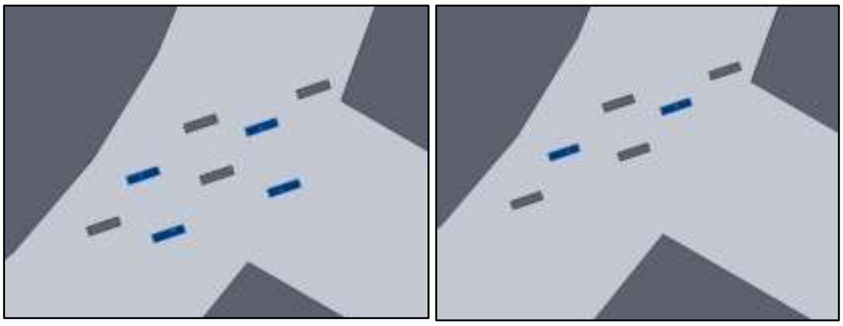

**Figure 8. Increase in the number of the vanes from 4 to 6 and from 4 to 8 for the positioning angle of 60 degrees.**

In order to investigate the flow condition in the lateral canal and the discharge to the diversion canal for the 6 and 8 submerged vanes, numerical simulation is performed. The corresponding relative discharge to the lateral intake canal for both models and for various Froude numbers are extracted. The Froude number values of the flow in the main canal are 0.21, 0.26, 0.31 and 0.37. In the diagram of the Figure 9. comparison is made for the 4, 6 and 8 submerged vanes and per various Froude number values.

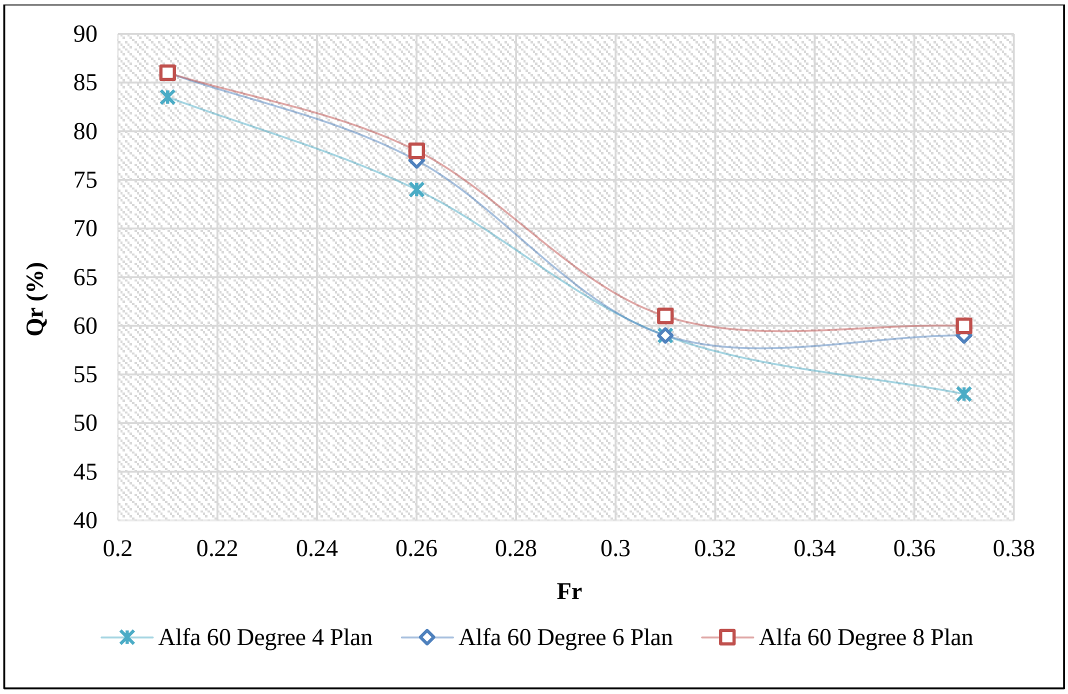


**Figure 9. Comparison between relative discharge values in the lateral canal with increase in the number of the submerged vanes and for the various Froude number values**

As seen in the Figure 9, the relative discharge values in the lateral canal increase with increase in the number of the submerged vanes from 4 to 6. Further increase in the number of vanes from 6 to 8, does not result into considerable increase in the relative discharge values and this increase has not a significant effect. Comparing between mean relative discharge in the lateral canal with increasing the number of submerged vanes shows that the case of 6 submerged vanes causes an increase about 6.1%in the discharge values. The reason for this is reduction in the local flow velocity at the entrance zone and juncture of the lateral canal and the main canal. Although increasing the number of vanes from 6 to 8 causes no further blockage of flow. It should be noted that increasing too much the number of submerged vanes at the juncture could result into reduced discharge due to the blockage of the lateral canal entrance. In the second part of this section for the case of 6 submerged vanes and positioning angle of 60 degrees which has up to now yielded the maximum relative diverted discharge, various arrangements are recommended. In these arrangements attempt is made that by appropriate diversion of the flow lines towards the lateral canal, the best results could be achieved. In the Figure 10, 3 different arrangements are introduced for the submerged vanes which are considered in the performed modelling per different Froude number values.

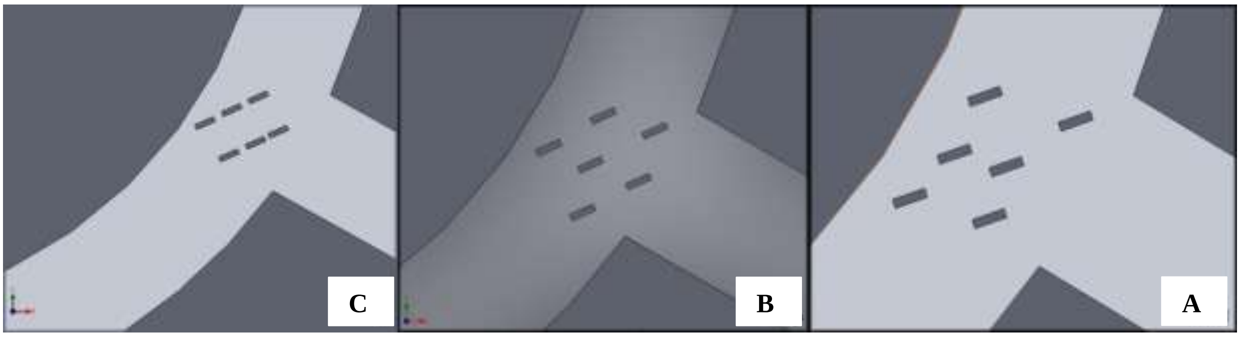


**Figure 10. A-Irregular, B-plaid, C-parallel arrangements of the submerged vanes at the juncture of the lateral canal and the main flow canal**

As seen in the Figure 10, the flow field in the lateral intake canal would be investigated with three different arrangements. In the first type arrangement, the submerged vanes are positioned irregularly at the lateral canal juncture and with an angle of 60 degrees. In the second type arrangement the vanes are positioned as plaid at the juncture. In the third type arrangement also 6 vanes are positioned parallel to each other in two rows at the lateral canal juncture. To investigate each arrangement the numerical model is simulated with 4 different Froude number values and the results are compared to each other. In the diagrams of the below figure the diverted discharge for each of the arrangements is shown.

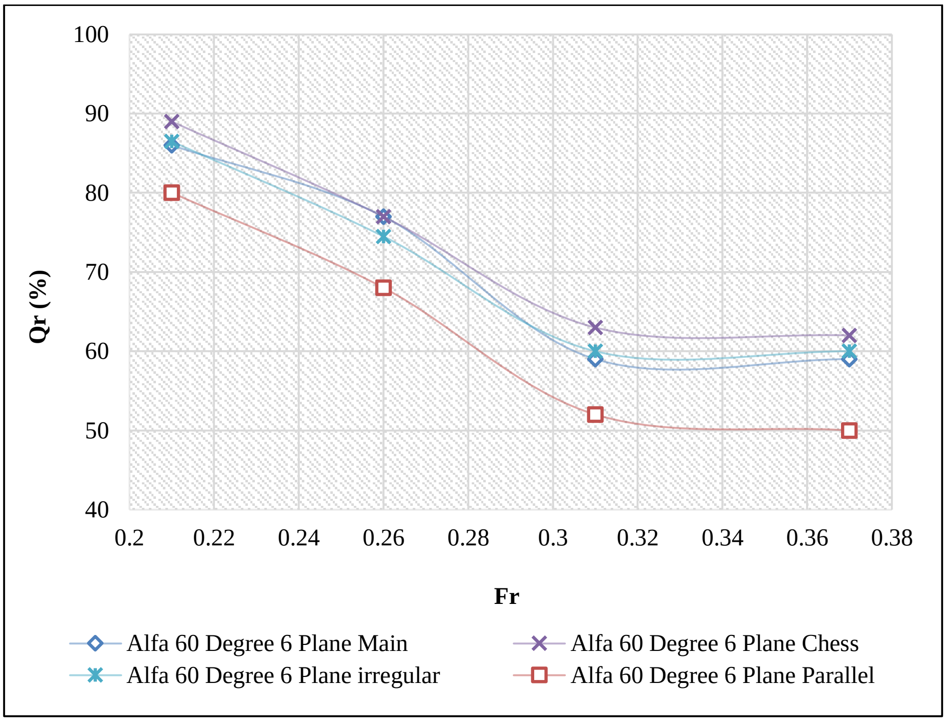


**Figure 11. Comparison between the relative discharge values in the lateral canal with different submerged vanes arrangements**

The results of various submerged vanes arrangements in the Figure 11. show that the type of submerged vanes arrangement could be an effective parameter on the relative discharge values in the lateral canal. So that parallel positioning of the vanes with 60 degrees angle, does not provide an appropriate condition for increase in the relative discharge value in the lateral canal. According to the numerical model results the plaid and random arrangements provide the best condition for the flow in the canal. In the plaid arrangement the flow is completely diverted toward the lateral canal and thus the discharge value is increased. By random arrangement of the submerged vanes along the flow path, the mean discharge in the lateral canal could be increased by 4%. The plaid arrangement of the vanes also increases the relative mean discharge value by about 4.7%. In this type of arrangement, increase in the flow turbulence in this zone causes reduced local velocity and diversion of the flow towards the lateral canal. In the plaid arrangement a greater width of the flow becomes turbulent. In the parallel arrangement, due to the insignificant effect of the layer parallel to the vanes on the flow, we would have practically no increased amount of discharge and this type of arrangement is not recommended for discharge enhancement in the lateral canal.

In the next step of the present study, the effect of submerged vanes' dimensions on the discharge in the lateral canal is investigated. For this purpose, using the results of the previous section for the plaid arrangement and the positioning angle of 60 degrees which diverts the maximum flow towards the lateral canal the submerged vanes dimensions are changed in two cases of increased and decreased dimensions. For this purpose in the first case the length, height and width of the block are reduced by 50% and in the next case they are increased by 50%. In the Figure 12 the positioning of the vanes is presented.

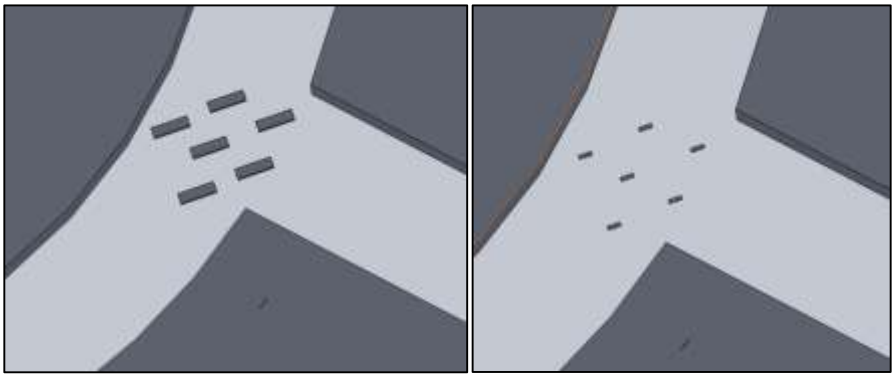

**Figure 12. The 50% increase and decrease in the submerged vanes dimensions for investigating the effect of vanes dimensions on the discharge into the lateral canal**

Based on the designed 3D geometries, the numerical modelling is performed for the conditions of 50% increase and decrease in the submerged vanes dimensions and the discharge values into the lateral canal in a 180 degrees bend are extracted. In the diagrams of the Figure 13. the relative discharge values in the lateral canal are extracted and drawn for the cases of 50% increase and decrease in the submerged vanes dimensions and per various Froude number values.

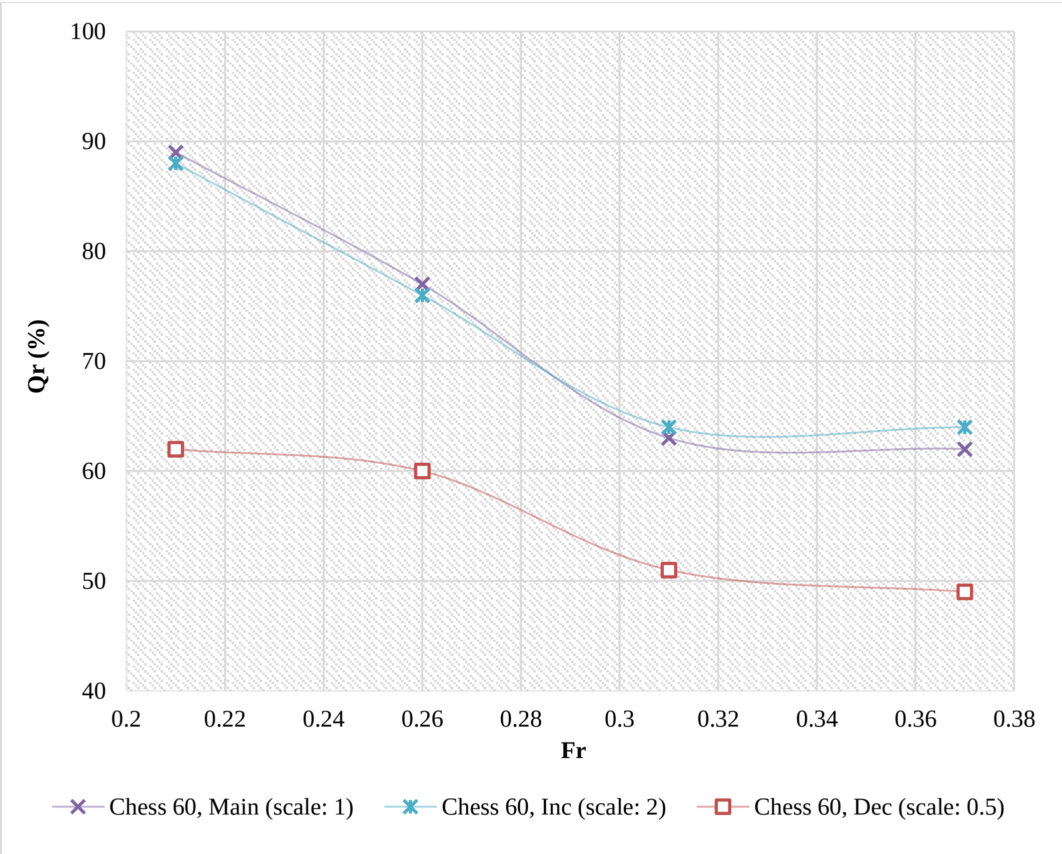


**Figure 13. Comparing the effects of 50% decrease and increase in the submerged vanes dimensions on the relative discharge values in the lateral canal within 180 degrees bend.**

As seen in the Figure 13. by 50% reduction in the submerged vanes dimensions, the relative discharge values in the lateral canal are greatly reduced. Examining the mean discharge values corresponding to all the Froude number values shows that with respect to the initial dimensions, the 50% reduction in the blocks' dimensions results in 31% reduction in the discharge values. Also by 50% increase in the blocks' dimensions, a significant increase is not observed in the relative discharge values of the lateral canal. So that the calculated mean relative discharge in the case of initial dimensions is increased only 1.5% with respect to the case of 50% increase in the blocks' dimensions. Therefore, based on the numerical analysis results, increase in the blocks dimensions would not result in significant increase in the relative discharge to the lateral canal. Because the conditions for maximum flow are provided in terms of the positioning angle, type of arrangement, number of vanes and block dimensions.

## 6. Conclusion

The submerged vanes with 10 degrees angle have the minimum relative diverted discharge and those with 60 degrees angle have the maximum relative diverted discharge. It could be concluded that with increase in the angle of submerged vanes in the canal bend, the value of relative diverted discharge or in other words the value of discharge in the lateral intake canal increases. The relative discharge values in the lateral canal increase with increase in the number of submerged vanes from 4 to 6. Further increase in the number of submerged vanes from 6 to 8, does not result in considerable increase in the relative discharge values and this increase has not a significant effect. Comparing the mean relative discharge in the lateral canal with increase in the number of submerged vanes shows that a set of 6 submerged vanes would increase discharge values about 6.1%. The corresponding results of the models with various arrangements of submerged vanes show that the type of submerged vanes arrangement could be an effective parameter on the relative discharge values in the lateral canal. So that parallel arrangement of vanes with 60 degrees angle does not provide appropriate conditions for increase in the relative discharge values in the lateral canal. According to the results of the numerical model, the plaid and linear arrangements provide the best condition for discharge in the canal. The plaid arrangement causes an increase about 3.7% in the mean relative discharge of the lateral canal. In this arrangement, increase in the flow turbulence in this zone causes local decrease in the velocity and diversion towards the lateral canal. In the plaid arrangement a greater width of the turbulent flow is formed. In the parallel arrangement due to negligible effect of the flow layer parallel to the vanes, we have practically no increase in the discharge value and these types of arrangements are not recommended for increasing flow in the lateral canal.

## 7. References


[1] Odgaard, A. Jacob, and John F. Kennedy. "River‐Bend Bank Protection by Submerged Vanes." Journal of Hydraulic Engineering 109, no. 8 (August 1983): 1161‐1173. doi:10.1061/(asce)0733-9429(1983)109:8(1161).

[2] Nakato, Tatsuaki, John F. Kennedy, and Donn Bauerly. "Pump‐Station Intake‐Shoaling Control with Submerged Vanes." Journal of Hydraulic Engineering 116, no. 1 (January 1990): 119‐128. doi:10.1061/(asce)0733-9429(1990)116:1(119).

[3] Wang, Yalin, and A. Jacob Odgaard. "Flow Control with Vorticity." Journal of Hydraulic Research 31, no. 4 (July 1993): 549–562. doi:10.1080/00221689309498877.

[4] Johnson, P. A., R. D. Hey, M. Tessier, and D. L. Rosgen. "Use of Vanes for Control of Scour at Vertical Wall Abutments." Journal of Hydraulic Engineering 127, no. 9 (September 2001): 772–778. doi:10.1061/(asce)0733-9429(2001)127:9(772).

[5] Barkdoll, Brian D. "Discussion of 'Use of Vanes for Control of Scour at Vertical Wall Abutments' by P. A. Johnson, R. D. Hey, M. Tessier, and D. L. Rosgen." Journal of Hydraulic Engineering 129, no. 3 (March 2003): 246–246. doi:10.1061/(asce)0733-9429(2003)129:3(246).

[6] Johnson, P. A., R. D. Hey, M. Tessier, and D. L. Rosgen. "Closure of 'Use of Vanes for Control of Scour at Vertical Wall Abutments' by P. A. Johnson, R. D. Hey, M. Tessier, and D. L. Rosgen." Journal of Hydraulic Engineering 129, no. 3 (March 2003): 247–247. doi:10.1061/(asce)0733-9429(2003)129:3(246.2).

[7] Michell, Frank, Robert Ettema, and Marian Muste. "Case Study: Sediment Control at Water Intake for Large Thermal-Power Station on a Small River." Journal of Hydraulic Engineering 132, no. 5 (May 2006): 440–449. doi:10.1061/(asce)0733-9429(2006)132:5(440).

[8] Borghei, S. Mahmood, and Aidin Jabbari Sahebari. "Local Scour at Open-Channel Junctions." Journal of Hydraulic Research 48, no. 4 (August 2010): 538–542. doi:10.1080/00221686.2010.492107.

[9] Sharma, H., Jain, B. and Ahmad, Z., 2016. Optimization of submerged vane parameters. Sādhanā, 41(3), pp.327-336.

[10] Ouyang, H., and C. Lu. "Optimizing the Spacing of Submerged Vanes Across Rivers for Stream Bank Protection at Channel Bends." Journal of Hydraulic Engineering 142, no. 12 (December 2016): 04016062. doi:10.1061/(asce)hy.1943-7900.0001210.

[11] Dey, Litan, Abdul Karim Barbhuiya, and Piya Biswas. "Experimental Study on Bank Erosion and Protection Using Submerged Vane Placed at an Optimum Angle in a 180° Laboratory Channel Bend." Geomorphology 283 (April 2017): 32–40. doi:10.1016/j.geomorph.2017.01.022.

[12] Barmaki, M., Fathiyan R., "Laboratory study of the effect of Submerged Vanes on the Lateral Intake Discharge Located in the 180 Degree Bend, 11th Iranian Hydraulic Conference, 1391, Orumieh, Iran.

[13] Yamini, O. Aminoroayaie, M.R. Kavianpour, and Azin Movahedi. "Pressure Distribution on the Bed of the Compound Flip Buckets." The Journal of Computational Multiphase Flows 7, no. 3 (September 2015): 181–194. doi:10.1260/1757-482x.7.3.181.

[14] Qi, Lan, Hui Chen, Xiao Wang, Wencai Fei, and Donghai Liu. "Establishment and Application of Three-Dimensional Realistic River Terrain in the Numerical Modeling of Flow over Spillways." Water Science and Technology: Water Supply (May 31, 2017): ws2017101. doi:10.2166/ws.2017.101.