# Numerical Modelling of Flow towards Headwork using CFD Approach

**Agraj Khakurel[1], Dr. Santosh Bhattarai[2], Dr. Bhola Nath Sharma Ghimire[3],**

**Dr. Ram Krishna Regmi[4]**

*[1]M.Sc. in Hydropower Engineering, Department of Civil Engineering, IOE, TU, Lalitpur, Nepal.*
*[2,4]Assistant professor, M.Sc. in Hydropower Engineering, Department of Civil Engineering, IOE, TU, Lalitpur, Nepal.*
*[3]Professor, M.Sc. in Hydropower Engineering, Department of Civil Engineering, IOE, TU, Lalitpur, Nepal.*

---***---

**Abstract -** *Headwork design play a vital role in the operational period of the project as these are components designed to divert a calm water flow and reduce the sediment entering into the intake as much as possible. In this study, hydrodynamic modelling of Solu Khola Dudhkoshi Hydroelectric project (SKDHEP) is done using three-dimensional (3D) computational fluid dynamics (CFD) software FLOW-3D for the final recommended design case (RDC) during physical modelling. A free surface flow with water as a sole fluid is used for the simulation. Considering gravity, viscosity and turbulence effect, the model uses volume flow rate as inlet boundary condition and outflow at the outlet with void initial condition in both conditions. Further hydraulics of flow was modelled changing intake orientation and changing the shape of divide wall of the RDC case. The flow hydraulics for RDC was found to be better as compared to modified cases for the same flood discharge. The flow contained vortices in front of modified intake while velocity was found to be higher for modified divide wall case. Additionally, coefficient of determination ($R^2$) value for RDC is obtained during the correctness test of the models by establishing correlation between flow parameters with physical model results for various flow discharges in the river.*

***Key Words***: **FLOW-3D, Void condition, Viscosity and turbulence, Hydraulics, Intake orientation, Divide wall**

## 1. INTRODUCTION

Nepal holds abundance of fresh water, originating from the Himalayas as snow fed rivers in the form of free surface runoff. So, proper and optimum utilization of water resource plays an important role in the economic prosperity of the nation. However, this task is not as simple. Many factors like hydrology, hydraulics and sediment and their interactions plays a major role for the successful construction and operation of a hydropower. The run of river plants are usually constructed in high mountainous regions due to availability of high head. But the flows in such area may be turbulent and have very high velocity due to steepness of the topography. Hence, proper study of the hydrodynamics and sediment is important for the efficient running of hydropower plants.

The design of an intake has traditionally been refined by carrying out physical model studies. However, with advancements made on computing, the complex flow problems can easily be solved by the use of Computational fluid dynamics (CFD) algorithms, which simplifies the full Navier Stokes equation for solving the fluid flows [1], [2]. In case of run-of-river (RoR) plants, the numerical modelling is not confined only to analyze the hydraulics of headwork components but also to optimize their design as well as the effect of those components on the flow sequence at the headwork region [3].

Study of flow hydraulics in the real environment itself is a difficult task as it involves number of complex phenomena. Similarly, a physical model study can take longer time and larger space to setup and is very difficult to measure flow properties. For these reasons, numerical modelling may be a handy solution to overcome all those problems related to hydrodynamic study [4]. Numerical model can be used to study the processes without missing out on smaller details that may not be visible manually in a physical model or even in a prototype [5]. Without validation of numerical model results with physical model results, numerical modelling seems doubted. Hence, the use of numerical modelling to simulate the phenomenon and verification of its result with physical modelling results, may prove to be fruitful and hence, can be used to study flow properties in much detail [1].

Numerous research has been carried out to model the flow hydraulics in various fields using different numerical models and techniques. Flood simulations by numerical modelling approach to help mitigate the flood damages before its occurrence and hydrological and hydrodynamic modelling for predicting the discharges at Brahmani-Baitarani river basin was carried out using CARTOSAT DEM and Indian Remote Sensing in HEC-HMS, HEC-GeoHMS, HEC-RAS and HEC-GeoRAS by Sindhu and Durga Rao (2016) [6]. Sandbach et al. (2018) applied 2D and 3D models in Delft3D to simulate the flow at the transition between river and estuary of the Columbia River Estuary [7]. Herrera-Granados and Kostecki (2016) applied 2D and 3D modelling for river and ogee weir over a Niedów barrage using SMS-FESWMS model and Flow3D respectively to determine flow pattern over the Niedów barrage located at South Poland. They also carried out physical modelling to compare the discharge coefficient values and concluded that the results are in agreement [8].

In this study, attempt has been made to simulate the flow hydrodynamics of SKDHEP using commercial CFD software Flow-3D.

## 2. PROJECT DESCRIPTION AND DATA COLLECTION

SKDHEP located in Solukhumbu district of Sagarmatha zone in Nepal is R-O-R type of project utilizing flow from Solu River to generate electricity. Location map of the project is shown in figure 1. It's a snow fed river with a catchment area of 454 km$^2$ at the headwork site. The headwork is located at the Solu River, having gradient of 1 in 15 while the powerhouse is located at right bank of Dudhkoshi River. The project has a net head of 598.09 m resulting in the installed capacity of 86 MW. Weir crest elevation is fixed at 1262m amsl and intake crest elevation was fixed at 1258m amsl. Three numbers of undersluices with two of sizes 1.5m X 1 m and one of size 1.25m X 1m are proposed [9], [10]. Figure 2 shows the confluence of Solu River and Dudhkoshi.

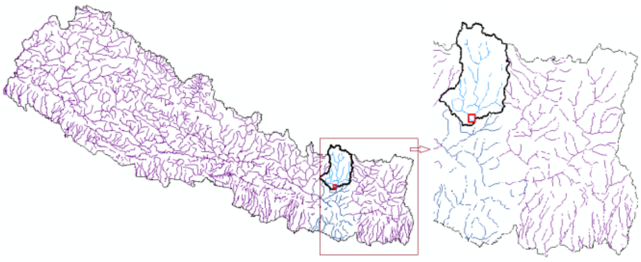

**Fig -1**: Location Map of SKDHEP

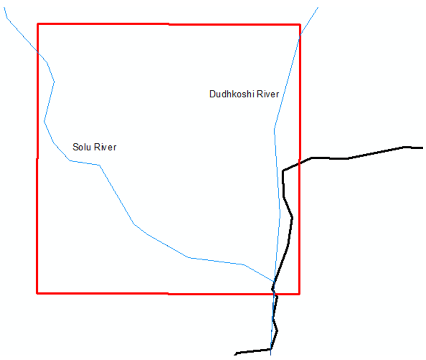


**Fig -2**: Detailed view of rivers at vicinity of project

The physical model study of SKDHEP was carried out on the river model as an existing situation test (EST) and on the headwork of the base case by Hydro Lab Pvt. Ltd. Based on hydraulic study, the base case design of headwork was modified and final recommended design case (RDC) was proposed [9]. The necessary survey data and drawings for RDC along with physical model report of the project was obtained from Sahas Urja Ltd.

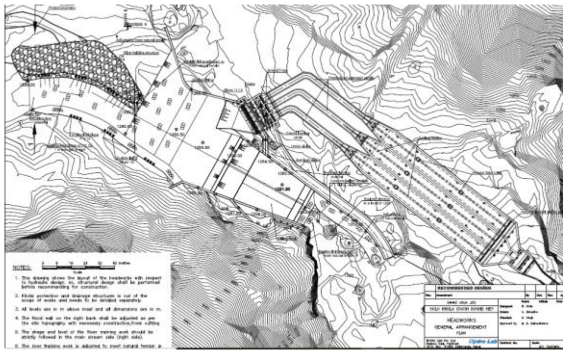

**Fig -3**: Headwork Arrangement for RDC (Hydro Lab Pvt. Ltd., 2018)

## 3. METHODOLOGY

In this study, the model for numerical approach was based on recommended design case. Firstly, the model for the headwork was built in AutoCAD. The model was then converted to stereolithography (.stl) format as FLOW-3D accepts to import geometry in this format.

Flow3D is a powerful, leading commercial package used for CFD modelling by approximating flow equations for three dimensional complex fluid flows. The domain is divided into a number of finite volumes and the equations that govern the flow is applied for each volumes within the domain. Flow3D is particularly useful to visualize free surface flows and flow processes occurring in a physical environment [11]. As materials properties, such as density, velocity and pressure varies considerably within the domain in the free surface flows, the simulation of free surface flows are not quite easy. However, in FLOW-3D, Volume of Fluid (VOF), developed by Hirt et al. (1975) is used which perform quite well when it comes to free surface flows. The VOF model is further refined and modified into TruVOF which is a leading algorithm for the simulation of free surface flows. In VOF model, volume of fluid function is first defined, a method to solve the VOF transport equation is set and the boundary conditions at the free surface is assigned [5].

### 3.1 Model Setup

After the completion of three-dimensional model, it was imported to Flow-3D in stereolithography (stl) format. In the general tab, the finish time for simulation was set and additional condition of total mass, average mean kinetic energy, average mean turbulent energy and average mean turbulent dissipation as steady parameters were selected for steady watch list [12]. A single fluid with incompressible flow mode and free surface interface tracking was selected for simulation.

### 3.2 Meshing and Grid Independence check

For the meshing of the imported geometry, the grid independence check was performed ranging from coarser (1.25 m) grid to finer mesh (0.75 m), along with another mesh block of 0.5m grid at intake for hundred year return period flood. The number of cells in the domain and time required for the simulation to complete are shown in the table below.

**Table -1:** Grid size characteristics for simulation

| Mesh Type | Coarser | Medium | Finer |
|---|---|---|---|
| Number of mesh blocks | 2 | 2 | 2 |
| Grid Size (m) | 1.25 | 1 | 0.75 |
| Total number of cells | 789,122 | 1,505,542 | 3,444,206 |
| Real simulation time | 343 sec | 543 sec | 343 sec |

The velocity contour for all three simulations were observed to be similar. Any of the above mesh can be adopted for the study. However, when finer mesh is used, more computational effort is used. Also, for coarser mesh, the simulation completes faster. So, based on computational effort as well as time for simulation (CPU time), the simulation with mesh size of 1m was adopted and carried out for twenty year return period flood too.

### 3.3 Fluid and Physics Definition

The fluid for the simulation was loaded from the fluid database as water at 20°C and the gravity was set to be $9.81 m/s^2$ downwards in Z-direction and viscosity and turbulence model was set as Renormalization Group (RNG) model.

### 3.4 Boundary and Initial Conditions

At the inlet of domain, Volume flow rate boundary condition was set. The water levels at inlet was computed from HEC RAS 5.0.5 and was also input to the model of corresponding discharges as shown in table 2. At the outlet, the outflow boundary condition was used. Other boundaries were defined with symmetry B.C. As an initial condition, the domain was not loaded with water i.e. void initial condition was set.

**Table -2:** Water Levels at inlet of model for various discharges using HEC RAS 5.0.5

| Discharge ($m^3/s$) | Water Level (m) | Remarks |
|---|---|---|
| 145 | 1271.79 | Annual Flood |
| 256 | 1272.04 | 1 in 5 years flood |
| 362 | 1272.19 | 1 in 20 years flood |
| 475 | 1272.32 | 1 in 100 years flood |
| 587 | 1272.43 | 1 in 500 years flood |

### 3.5 Simulation Run

The numerical model was run for two floods of return periods 20 years and 100 years. Under the numerics tab, the advection was taken to be implicit. The total number of computational meshes of numerical model for these floods in RDC were 1,505,542 for both 20 years and 100 years flood. The velocities obtained at different sections were then compared to that of physical model study report.

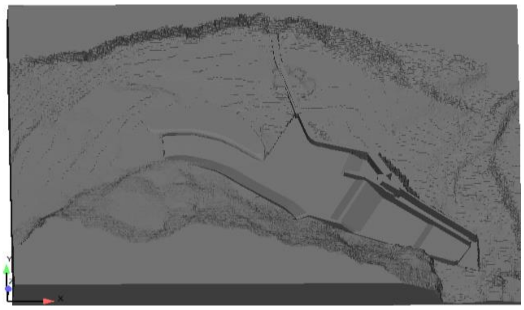

**Fig -4**: 3-D Geometry Model (RDC)

Finally, the model was modified by changing the intake orientation (perpendicular to the flow) in one case and changing the divide wall as another case. The same procedure was carried out for simulation considering the design flood of 475 $m^3/s$. After the simulation became steady, the flow hydraulics for these cases were analyzed and compared to the RDC case for same discharge over the domain.

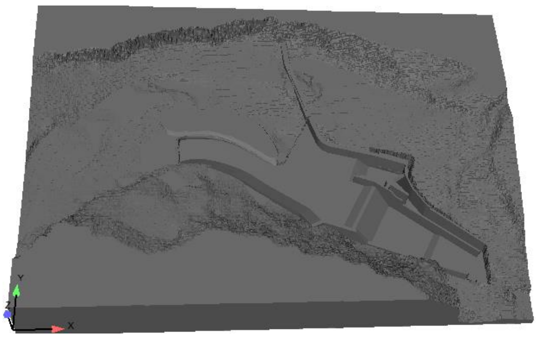

**Fig -5**: 3-D Geometry Model (Modified Intake Orientation)

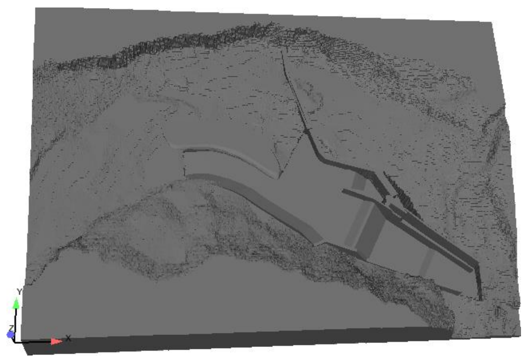

**Fig -6**: 3-D Geometry Model (Modified Intake Orientation)

## 4. RESULTS AND DISCUSSION

The numerical results were validated with the experimental results using FLOW-3D model using CFD approach. The RDC model was then modified by changing the intake orientation to obtain the hydrodynamics in the headwork considering all other conditions remaining same. Finally, RDC model was modified by changing the divide wall orientation to compare the flow pattern with that of RDC for a flood of 100 years return period.

For the purpose of validation, the numerical model (N.M.) results were compared to physical model (P.M.) results by Hydro Lab Pvt. Ltd. (2018) at various sections within the domain. The velocities were measured along the thalweg at various chainage of the RDC model. Table 3 below shows the velocities measured for different discharges along thalweg for the chainages in which velocity measurement was carried out in experimental study. The river is highly turbulent and steep with gradient of 1:15 [9].

**Table -3:** Comparison of velocities for physical and numerical models

| Chainage (m) | Velocity along Thalweg (m/s) | | | |
|---|---|---|---|---|
| | 20 year return period flood | | 100 year return period flood | |
| | P.M. | N.M. | P.M. | N.M. |
| 180 | 6.51 | 6.659 | 8.01 | 8.161 |
| 220 | 6.49 | 6.281 | 6.07 | 5.989 |
| 383 | 2.13 | 2.058 | 5 | 5.064 |
| 393 | 1.75 | 1.706 | 2.97 | 2.979 |
| 408 | 2.99 | 2.554 | 3.23 | 3.182 |

The figures below shows the flow pattern from physical and numerical models and water surface profile obtained from the numerical model for RDC cases for discharges of twenty and hundred years return periods.

**Fig -7**: Flow pattern for RDC for 20 year flood (Hydro Lab Pvt. Ltd., 2018)

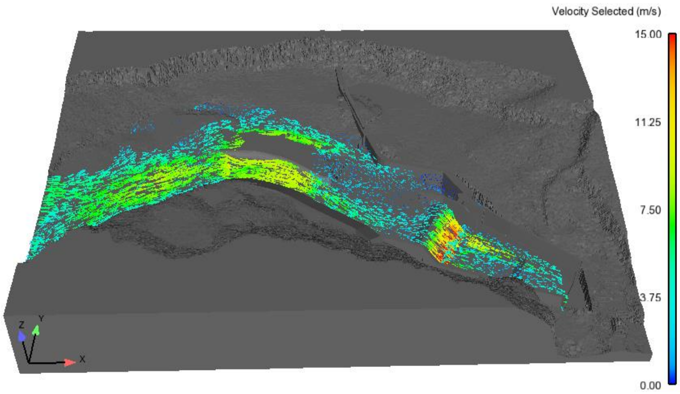


**Fig -8**: Flow pattern for RDC for 20 year flood (numerical model)

**Fig -9**: Flow pattern for RDC for 100 year flood (Hydro Lab Pvt. Ltd., 2018)

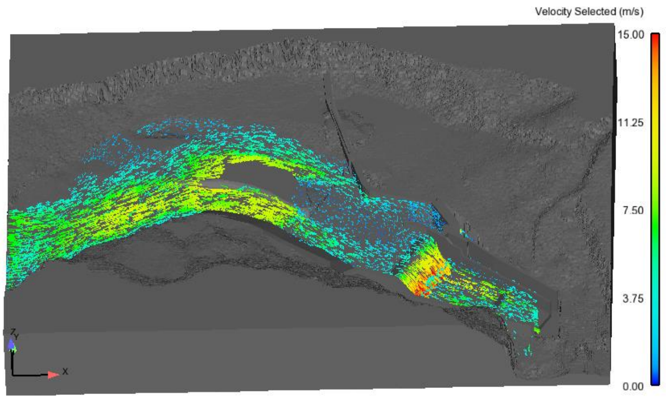


**Fig -10**: Flow pattern for RDC for 100 year flood (numerical model)

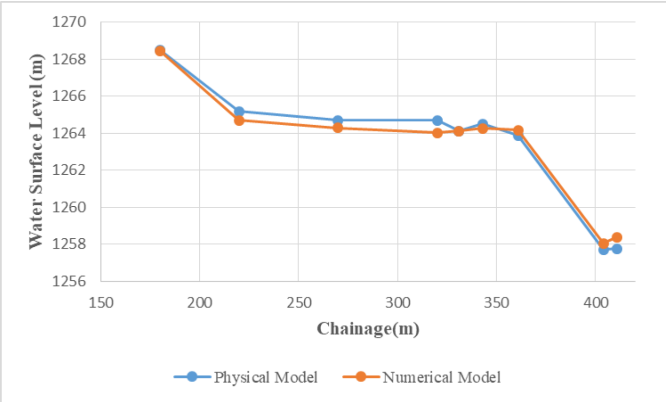


**Fig -11:** Water surface profile for RDC for 20 year flood

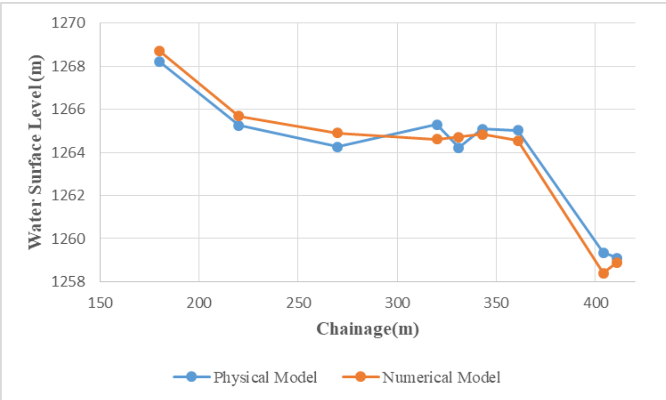


**Fig -12:** Water surface profile for RDC for 100 year flood

Due to the river training before the headwork, the flow was distributed towards left and right banks. The flow upstream of weir crest at the middle was found to be turbulent. The wave formation was also found in physical model at the mid upstream of weir crest in physical model. The flow from the left bank was found to divide towards mid portion of the floor upstream of weir crest as well as towards intake.

There may be various reasons for differences in flow parameters from experiment and numerical model. The comparison between numerical and physical model velocities at various sections was found to be within 5% except for the last chainage in 20 year flood where deviation was found to be within 15%. This may be due to assumptions in physical model study, presence of flow vortices, turbulence and complexities arising in the flow itself [13].

In order to check the correctness of the model, the coefficient of determination ($R^2$) was determined for both flow velocity as well as water surface level and its value for each of the model was observed to be as in table below.

**Table -4:** Coefficient of determination for test cases

| Test for flood discharge of | Coefficient of Determination ($R^2$) | |
|---|---|---|
| | Velocity | Water Level |
| 20 year return period | 0.992 | 0.99 |
| 100 year return period | 0.998 | 0.975 |

From Table 4, it can be seen that the coefficient of determination is very high. Hence, a good correlation exists between the experimental and numerical model and hence, the numerical model is acceptable.

Next, the recommended design case was to be modified with a view to improve the hydraulic characteristics at the intake as according to physical model report, hydraulic performance of intake was not much satisfactory. Even after RDC was modelled, weak rotational flows were encountered [9]. Hence, the intake orientation and divide wall was modified to compare the changes in hydraulics between modified scenarios and RDC.

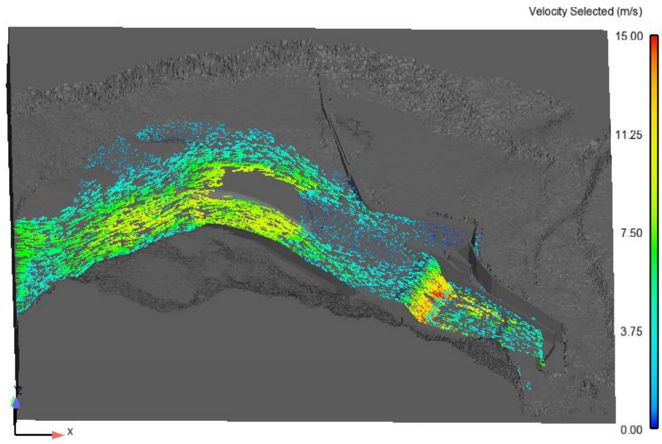


**Fig -13**: Flow pattern for Modified Intake Orientation case for 100 year flood (numerical model)

In Fig. 13, it was observed that the flow available in front of intake increased but the flow was not free from vortices at the intake pond. It is undesirable to have flow vortex in front of intake as the water should be calm and relatively sediment free. But when vortices occur, it does not allow the sediment to settle outside the intake orifice resulting in entry of sediments to the conveyance system.

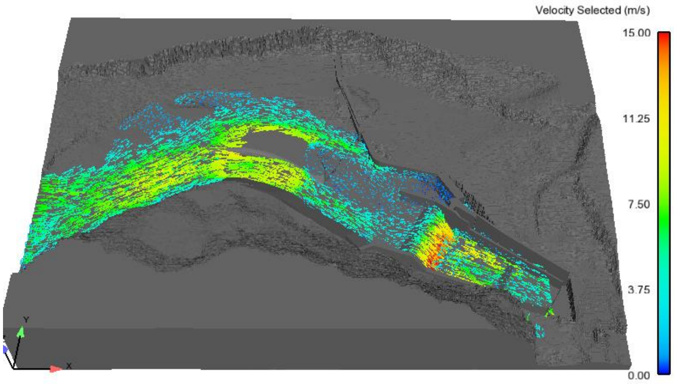


**Fig -14**: Flow pattern for Modified Divide Wall case for 100 year flood (numerical model)

Finally, in order to divide flow a bit upstream of intake with the expectation of obtaining a smooth vortex free water at intake pond, the divide wall was extended upstream and orientation was changed. However, in Fig. 14, it was observed that the flow pattern was similar to that of RDC for same flood throughout the domain except for the right side of the modified divide wall where small vortex formation was encountered. Also, most of the flow through the left bank was found to divert towards weir crest. The small rotational flows were still observed at intake pond.

## 5. CONCLUSION

In this paper, the flow parameters and flow pattern study of SKDHEP is carried out at the headwork. The flow parameters obtained from experiment for various discharges at various sections for RDC is verified and the accuracy of the model applied is tested. Since the coefficient of determination was sufficiently high, the numerical model is deemed to be acceptable.

In this way, a numerical model can be applied to determine the hydrodynamics of flow when the initial and boundary conditions are properly applied to the model using CFD approach. Also, the geometry can be modified to compare the hydrodynamics and obtain a better arrangement of headwork component for a hydropower system. From this study, a conclusion can be drawn that the modified scenarios could not improve the flow hydraulics as vortices were encountered in modified intake orientation case and uneven flow division as well as flow vortices were formed at right side of modified divide wall.

## ACKNOWLEDGEMENT

The authors are grateful to **Hydro Lab Pvt. Ltd.** for carrying out the physical model study of SKDHEP which is used as reference for numerical model study initially. The authors would like to acknowledge **Sahas Urja Ltd.** for providing the necessary survey data and drawings along with physical hydraulic model study report carried out on SKDHEP by **Hydro Lab Pvt. Ltd.** in 2018. The authors would like to express their deepest gratitude to **Flow Science, Inc.** for providing with an academic license of **FLOW-3D v12.0** without which the study would have been incomplete. Finally, the authors are grateful to **Er. Rajesh Bhattarai** for helping with the geometry modelling.

## REFERENCES

[1] M. C. Aydin and E. Isik, "Using CFD in Hydraulic Structures," International Journal of Scientific and Technological Research, 2015.

[2] Minnesota Department of Transportation, "Applications of Computational Fluid Dynamics for River Simulation: State of the Practice," Transportation Research Synthesis, 2013.

[3] V. Martin, "Hydrodynamic Modelling of Intakes for Run-of-River Hydroelectric Projects," 22nd Canadian Hydrotechnical Conference, 2015.

[4] S. Heimsund, E. W. Hansen and W. Nemec, "Computational 3-D fluid-dynamics model for sediment transport, erosion and deposition by turbidity currents," IAS 16th International Sedimentological Congress, 2002.

[5] Flow Science, Inc., "FLOW-3D® Version 12.0 Users Manual," Santa Fe, NM, USA, 2018.

[6] K. Sindhu and K. D. Rao, "Hydrological and Hydrodynamic Modeling for Flood Damage Mitigation in Brahmani-Baitarani River Basin, India," Geocarto International, 2016.

[7] S. D. Sandbach, A. P. Nicholas, P. J. Ashworth, J. L. Best, C. E. Keevil, D. R. Parsons, E. W. Prokocki and C. J. Simpson, "Hydrodynamic modelling of tidal-fluvial flows in a large river estuary," Elsevier, 2018.

[8] O. Herrera-Granados and S. W. Kostecki, "Numerical and physical modeling of water flow over the ogee weir of the new Niedów barrage," J. Hydrol. Hydromech., 2016.

[9] Hydro Lab Pvt. Ltd., "Physical Hydraulic Model Study of the Headworks of Solu Khola (Dudh koshi) Hydroelectric Project (SKDHEP)," Kathmandu, 2018, unpublished.

[10] Sahas Urja, "Sahas Urja," 2019. [Online]. Available: https://sahasurja.com/project/solu-khola-dudhkoshi-hydro-electric-project-86mw/. [Last accessed 29 March 2020].

[11] M. Taghavi and H. Ghodousi, "Simulation of Flow Suspended Load in Weirs by Using Flow3D Model," Civil Engineering Journal, vol. 1, 2015.

[12] A. S. Abdurrasheed, K. W. Yusof, E. H. H. Alqadami, H. Takaijudin, A. A. Ghani, M. M. Muhammad, A. T. Sholagberu, M. K. Zainalfikry, M. Osman and M. S. Patel, "Modelling of Flow Parameters through Subsurface Drainage Modules for Application in BIOECODS," Water, 2019.

[13] L. Choufu, S. Abbasi, H. Pourshahbaz, P. Taghvaei and S. Tfwala, "Investigation of Flow, Erosion, and Sedimentation Pattern around Varied Groynes under Different Hydraulic and Geometric Conditions: A Numerical Study," Water, 2019.