Investigation of Sediment Transport Through a Run-of-River Hydroelectric Project

Violeta Martin, Ph.D., P.Eng. Senior Hydrotechnical Engineer, Knight Piésold Ltd., Canada

Michael Pullinger, M.Sc., P.Eng., Project Engineer, Knight Piésold Ltd., Canada

Alana Shewan, M.A.Sc., P.Eng., Project Engineer, Knight Piésold Ltd., Canada

Nathan Smith, P.Eng., Project Engineer, Knight Piésold Ltd., Canada

Abstract

A 25 MW run-of-river hydroelectric project is proposed in coastal British Columbia, Canada. Baseline studies identified that the bedload sediment transport rate in the project area is relatively low, and consequently, interruption of bed material replenishment in the reaches downstream of the intake represents an environmental concern. A prerequisite for the successful design of the proposed project includes passing of the spawning size gravel through the intake facility during the period of headpond sediment infilling. A sluice gate has been incorporated in the intake structure to enable the sluicing of sediment. Numerical modelling was undertaken to aid with the intake design and to determine the sediment transport efficiency through the headpond. Hydrodynamic modelling techniques (1D, 2D and CFD) were combined to assess the proposed design and provide an overview of water depths, velocities and bed shear stresses along with the potential for gravel mobility through the headpond reach both before and after the construction of the intake.

1. Project Description

The proposed 25 MW run-of-river hydroelectric project (HEP) is located in coastal British Columbia, near Vancouver, Canada. The proposed intake structure is a concrete weir with a Coanda screen (Figure 1). The intake structure will capture and convey flows laterally into a water conveyance penstock down to the powerhouse. The intake structure will feature two sluice facilities located on either side of the Coanda screen. The sluice facilities will pass flows during spillway construction and operation and will facilitate the flushing of sediment through the headpond.

The sluice facility on the north side of the Coanda screen will act as the intake bypass during winter low flows and as a secondary sediment sluice gate. The sluice facility on the south side of the Coanda screen will be located in the deepest section of the headpond (former stream channel) and will be the primary sediment sluice gate to promote sediment flushing from the headpond to the downstream reach. This stream channel has been assessed to have a very low bedload sediment transport rate; therefore, the headpond was estimated to take many years, even decades, to infill and start passing bedload sediment over the Coanda screen. Proper functioning of the sediment sluice gate and passing of spawning size gravel through the intake is therefore, of high environmental value for this project.

An overflow weir with an Instream Flow Requirement (IFR) gate will be located next to the sediment sluice gate and will supply the required environmental flows to the downstream reach during normal operations. A construction diversion channel located on the south side of the intake structure will divert flows during construction. The construction diversion channel and berm will be left in place once operations commence and will be flooded within the headpond.

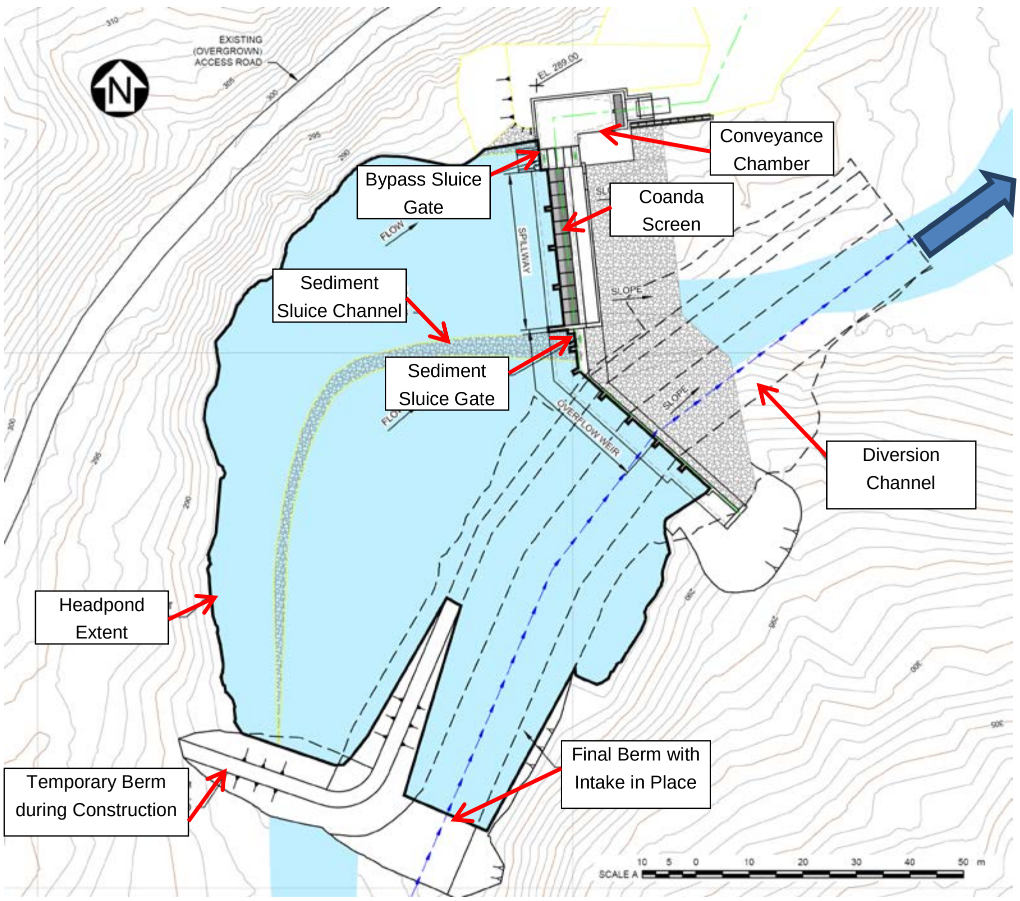

Figure 1: Intake Structure

2. Study Objectives and Modelling Scenarios

Numerical modelling was completed utilising a combination of 1D, 2D and CFD software packages. The numerical modelling software packages used included:

- One-dimensional: HEC-RAS
- Two-dimensional: FLO-2D, and
- CFD: ANSYS FLUENT.

The objective of the numerical modelling was to examine the hydrodynamics and sediment transport in the headpond and through the intake structure under various flow conditions. The flow conditions examined included operating conditions for the design discharge, sluicing conditions for a high annual daily discharge, and baseline sediment transport as follows:

- Scenario 1: Intake design discharge of 11.4 m^3/s with addition of minimum IFR of 0.6 m^3/s, (total flow of 12 m^3/s). This scenario was used to assess the flow and velocity patterns in the headpond for normal operations with no discharge through the sediment sluice channel (1D and CFD).
- Scenario 2: Intake design discharge of 11.4 m^3/s with addition of 12.6 m^3/s through the sediment sluice channel equivalent to a high annual daily discharge of 24 m^3/s. This scenario was used to assess the sediment mobility during normal power generation with a full headpond and with the sediment sluice channel open (1D and CFD).
- Scenario 3: High annual daily discharge of 24 m^3/s that is typical during the freshet season. This scenario was used to assess the effectiveness of the sediment sluice channel for sediment mobility with the sediment sluice gate open and the headpond lowered (i.e. all of the flow conveyed through the sluice channel; no flow through the Coanda screen intake) (1D and CFD).
- Scenario 4: Baseline conditions with the average monthly freshet flow in June (a streamflow of 12 m^3/s). This flow condition is equivalent to the flow tested in Scenario 1. This scenario was used to assess the sediment mobility in the existing stream channel prior to HEP construction (1D and 2D).
- Scenario 5: Baseline conditions with a streamflow of 24 m^3/s. This scenario was used to assess the sediment mobility in the existing stream channel prior to HEP construction for a high annual daily discharge (1D and 2D). This flow rate is equivalent to the flow tested in Scenarios 2 and 3.

3. Modelling Methods

The inputs, assumptions and methodology for the three models used in this study are discussed in the following sections. The extent of each model is shown on Figure 2.

The 1D and CFD models were used primarily for flow characterization and for hydrodynamic studies, while the 2D model was used for the baseline sediment transport studies. A 2D model was used to characterize the baseline sediment transport as a longer stream reach was necessary for a proper assessment through the area of interest. The variation of velocities through the depth was not essential in this case either, and hence CFD modelling was not required for this purpose. The 2D model results were used for comparison purposes and are therefore discussed in the final section of this paper.

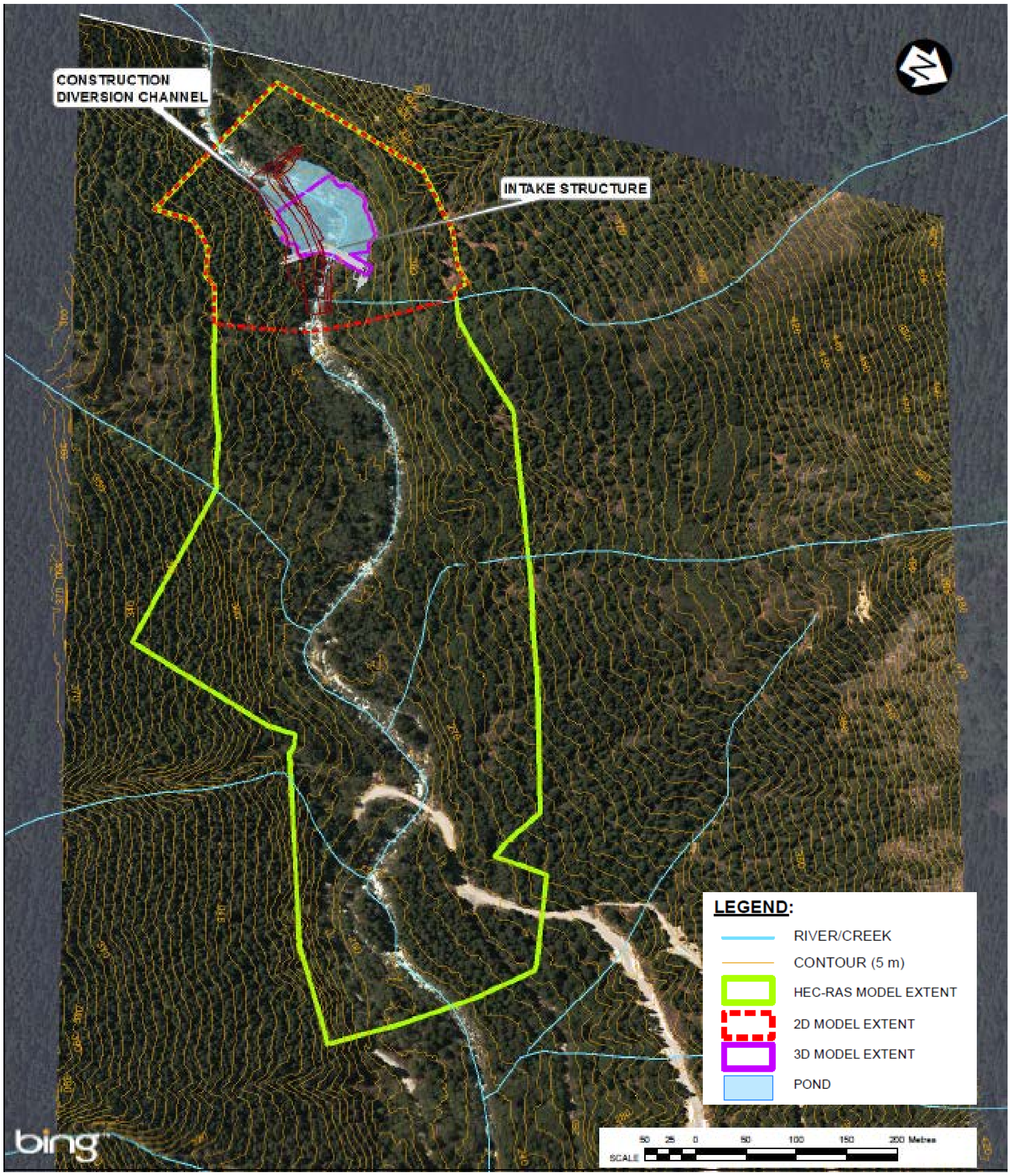

Figure 2: 1D, 2D and CFD Model Extents

3.1 One-Dimensional Model

The 1D model used for this study was HEC-RAS. HEC-RAS is a 1D hydraulic model developed by the U.S. Army Corps of Engineers Hydrologic Engineering Center and is primarily used to model water surface elevations for open channel flow. The HEC-RAS model was used in this study to assess the large scale hydraulics for the project, provide a calibration basis for the 2D and CFD models, and to determine the boundary conditions for the CFD model.

Inputs to the HEC-RAS model included channel geometry (cross-sections) based on 1 m LiDAR information for the project area. The cross-sections were placed to best represent the hydraulic conditions for both the existing and developed channel.

The HEC-RAS model was calibrated for baseline conditions using the developed rating curve for a gauging station installed approximately 500 m downstream of the intake location. The channel and overbank roughness parameters were adjusted in HEC-RAS until the modelled and measured water surface elevations were comparable. The calibration results are shown on Figure 3. For the range of flows investigated in Scenarios 1 through 5 (i.e. between 10 m^3/s and 30 m^3/s), the model was considered to be well calibrated. However, the capacity of the channel to convey low flows was somewhat underestimated in the HEC-RAS model, therefore resulting in higher simulated water surface elevations during low flows. This was because the geometry used for the model was based on 1 m LiDAR data and the topography below the water surface was not accurately captured for shallow flows.

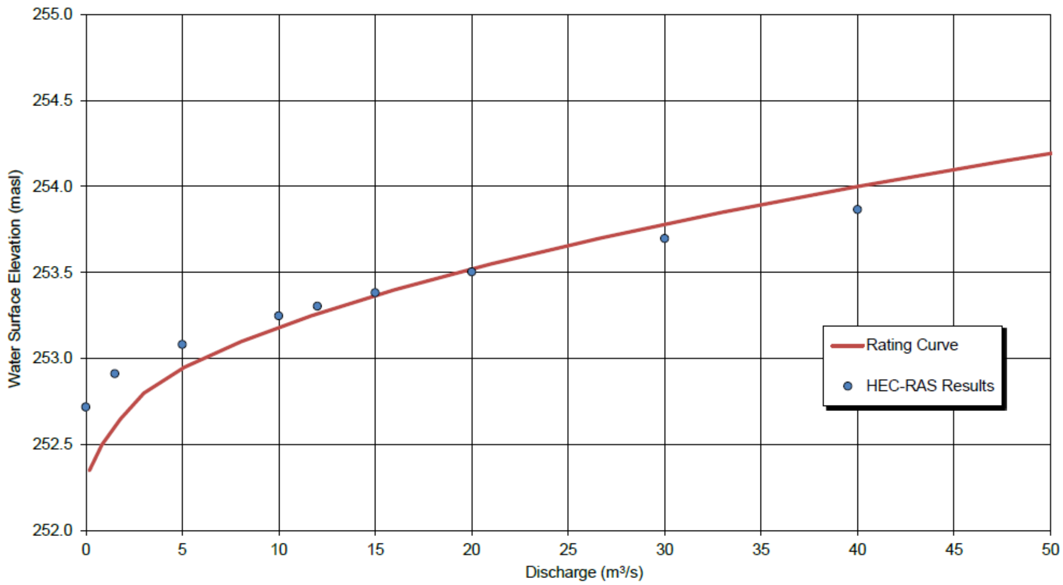

Figure 3: HEC-RAS model calibration

3.2 CFD Model

Computational Fluid Dynamics (CFD) model of the headpond and intake structure was created using ANSYS Fluent. This software is a general purpose CFD software with a potential to model a wide range of flow conditions. The strength of CFD modelling is in the ability to obtain flow conditions at any point in the flow field, including at various depths throughout the system. The purpose of this modelling was to investigate the overall hydrodynamics in the headpond, obtain estimates of bed shear stresses under various flow conditions, and evaluate the sediment mobility through the headpond.

Model Simplifications

The headpond and flow through the intake in Scenarios 1 and 2 were modelled as a single phase fluid with a zero shear stress wall boundary condition placed at the free water surface in the headpond. This simplification provides an accurate assessment of the flow field through the headpond, at the sluice gate, and along the river bed; however, it results in a small inaccuracy in the water surface elevation right at the weir crest during power generation. A more complex two-phase flow field was used for Scenario 3 due to the varying water surface elevation through the headpond region.

The upstream extent of the CFD model was defined at the limit of the headpond backwater extent during normal operations in Scenarios 1 and 2. The same model extent was used as the upstream boundary for Scenario 3. The intake structure defined the downstream extent of the CFD model and no modelling of flows downstream of the sluice gates or overflow weir were undertaken with the CFD model.

Numerical Grid and Solution Procedures

An unstructured tetrahedral mesh was created using the ANSYS meshing tool. Increased mesh fineness in the vicinity of the intake openings, sluice channel, and overflow weir was used to better define the hydraulics in these regions. The optimised mesh spacing resulted in a mesh with approximately 700,000 cells.

The realizable k-ε turbulence model and a steady state solver were utilised for Scenarios 1 and 2. Mass flow through the sluice gate was assessed for convergence and model stability was reached in approximately 3,000 iterations. Scenario 3 was run using the realizable k-ε turbulence model and a pseudo-transient solver. Mass flow through the sluice gate was assessed for convergence and model stability was reached in approximately 5,000 iterations.

Boundary Conditions

The boundary conditions used for CFD modelling were as follows:

- Mass flow inlet at the upstream boundary.
- Pressure outlet at each outflow, with outflow normal to the boundary. Scenarios with multiple pressure outlets had a target mass flow rate specified for all outlets.
- No-slip wall for the concrete wall of the intake structure, riverbed and riprap structures, with each surface having an appropriately defined roughness height that was fine-tuned in the calibration process.
- Wall with zero shear stress for the air boundary at the top of the domain.
- A “free surface” inlet boundary condition was defined for Scenario 3 to delineate the water level at the upstream boundary of the headpond (as taken from the 1D modelling results). The water surface throughout the remainder of the headpond was calculated by the software.

Sediment Mobility

For a given bed shear stress, the incipient motion of sediment particles was estimated using the Shields (1936) equation:

$$d_s = \frac{\tau}{\tau_0 g(\rho_s - \rho_w)}$$

d_s= sediment particle size that would have a potential to get mobilized (m)
$\tau =$ bed shear stress caused by the given flow condition (Pa)
$\tau_0 =$ non-dimensional critical shear stress equivalent to 0.06 for gravel/cobble/boulder bed stream channels
$\rho_s =$ sediment density (kg/m^3)
$\rho_w =$ water density (kg/m^3)

The CFD software calculates the bed shear stresses throughout the headpond as part of its calculation procedure (ANSYS, 2011). The sediment size that has a potential to get mobilized at any location in the headpond was then estimated based on the shear stress using the above equation. Deposition and scour of sediment are not directly simulated with the CFD software, but can be inferred from the above calculation in conjunction with the shear stress results.

The non-dimensional shear stress of 0.06 chosen in this study for evaluating the sediment size at incipient motion is typical for stream channels with gravel/cobble/boulder substrate material (Buffington and Montgomery, 1997; Martin, 2003). Local bed topography, however, plays an additional role in particle mobility and particles hiding in wakes of larger boulders or imbricated within the bed may remain immobile. Non-dimensional critical shear stresses encountered in complex channel morphologies were shown to be as high as 0.1 (Church et al., 1998). Consequently, the modelled results represent average conditions for the modelled reaches, whereas these conditions may differ locally depending on bed topography.

3.3 Two-Dimensional Model

The sediment transport capability of the natural stream channel upstream and downstream of the intake structure prior to construction was assessed for two flow conditions using the FLO-2D software package. The model extent is shown on Figure 2. A 2D model was selected for this assessment, because it was important to select a substantially larger area than in the CFD model to ensure that boundary effects had no impact on the results.

FLO-2D is a US Federal Emergency Management Agency (FEMA) approved hydraulic model for riverine studies and unconfined flood analyses. It is a two-dimensional flood routing model that simulates channel flow and unconfined overland flow over complex topography using either input hydrographs or rainfall-runoff simulations. The model uses the full dynamic wave momentum equation and a central finite difference routing scheme with eight potential flow directions (FLO-2D, 2012). The model contains sediment transport capabilities using one of nine available equations, where the sediment volume is conserved on a grid element basis.

Model Set-up and Calibration

The topography for the FLO-2D model was based on 1 m LiDAR data. The LiDAR was used to generate a continuous surface and a 2 m model grid was then set up in FLO-2D (Figure 4). Each grid element is represented by a single averaged elevation.

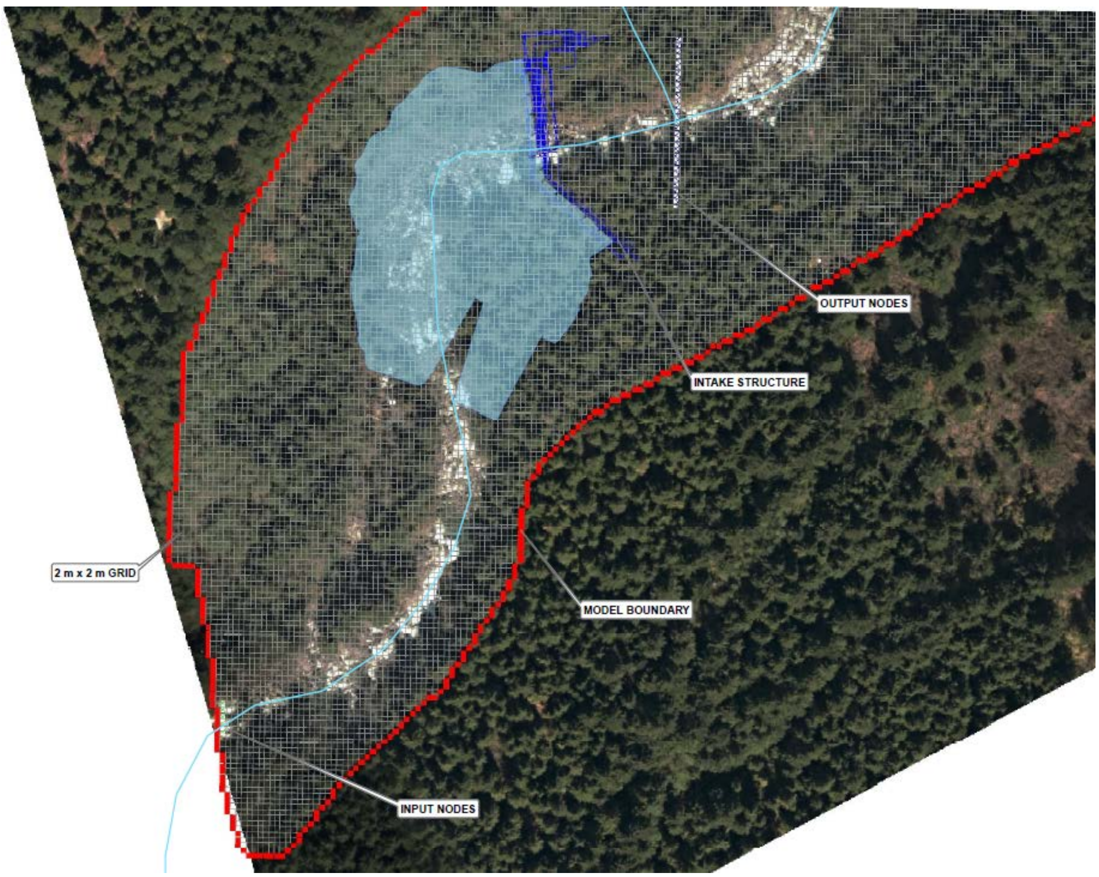

Figure 4: FLO-2D model grid with input and output nodes (intake structure and headpond extent shown for reference only)

Inflow to the FLO-2D model was uniformly spread over six grid elements to reduce numerical instability in the model. The elevations of the inflow nodes were arbitrarily increased to reduce potential ponding of flow at the model inflow boundary. The location of the inflow boundary was a sufficient distance upstream from the proposed intake structure location so that the flows were fully developed through the future headpond area and at the intake. Outflow in FLO-2D was created by adding outflow grid elements at the downstream boundary as shown on Figure 4. The model assumes a normal flow depth condition calculated based on the upstream grid elements. The location of the downstream outflow nodes was a reasonable distance away from the proposed intake location and had no impact on the flow patterns in the area of interest.

The model was calibrated by adjusting the Manning's roughness coefficient until the FLO-2D model results compared well with the HEC-RAS water surface elevations at various cross sections. The FLO-2D Manning's n value was typically higher than the value used in HEC-RAS. This discrepancy was expected as channel expansion, contraction and bend losses are taken into account through the Manning's n in FLO-2D while they are calculated as separate losses in HEC-RAS.

The Zeller-Fullerton (1983) equation was selected to model sediment transport in FLO-2D. This equation is a computer generated solution of the Meyer-Peter and Müller (1948) bed-load equation combined with Einstein's (1968) suspended load equation to generate bed load with no bed armouring (FLO-2D, 2012). For both flow scenarios a particle size of 50 mm was used to model sediment transport. This sediment size falls within the range of sediment sizes between 10 mm and 100 mm that are used by resident fish species for spawning (based on field studies at the project location).

4. 1D, 2D and CFD Modelling Results

The 1D and CFD models were used primarily for flow characterization during operating conditions (flow patterns, depths, velocities and shear stresses), while the 2D model was used to assess sediment mobility during baseline conditions for comparison with operating conditions. The 1D and CFD model results are described first, followed by the 2D model results at the end of this section.

4.1 One-Dimensional Model

The HEC-RAS results for the water surface profile for Scenarios 1, 2 and 3 are shown on Figure 5. These results were used as boundary conditions for the CFD model. Scenarios 1 and 2 represent operating conditions with power generation for the full design flow of 11.4 m^3/s. The headpond water surface elevations are the same for both of these scenarios as the same amount of water is passed over the Coanda screen as in Scenario 1, while the additional flow in Scenario 2 is passed through the sluice gate. The headpond extent for these two scenarios is the same as both conditions have the same water surface elevation.

Scenario 3 represents a lowered headpond with no power generation and all flow passing through the sluice gate. The modelling results for this Scenario (Figure 5) indicate that there is a backwater effect upstream of the sluice gate caused by the sluice opening being narrower than the natural stream channel upstream of it. The backwater effect extends approximately 30 m upstream of the intake structure.

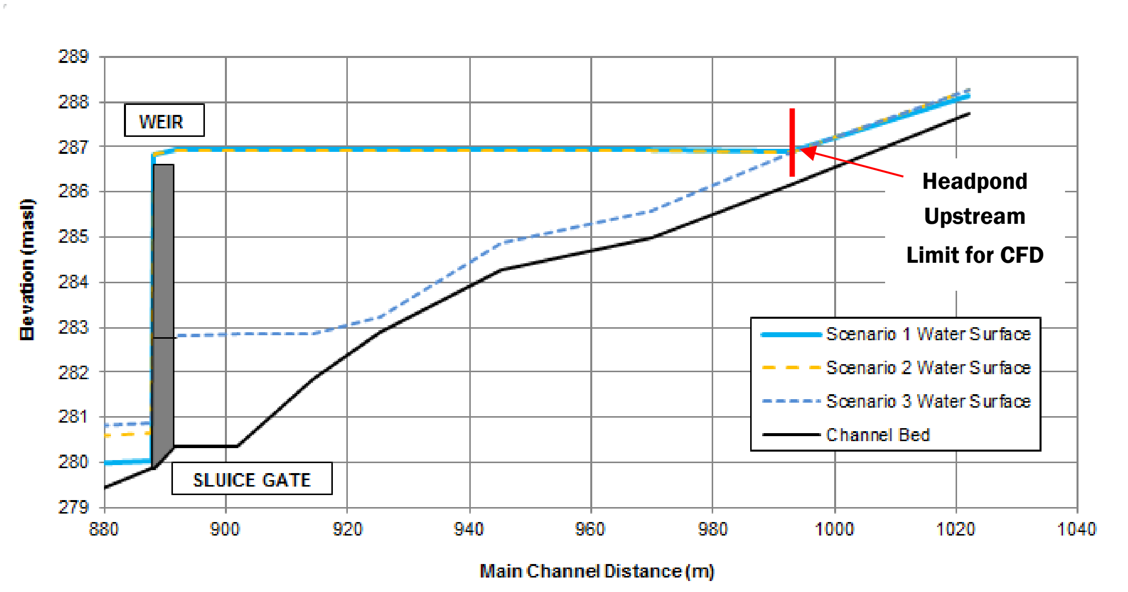

Figure 5: Water surface elevations for Scenarios 1, 2 and 3 calculated using HEC-RAS (also showing the upstream extent of the CFD model)

4.2 CFD Model

The results of CFD modelling for Scenarios 1, 2 and 3 are shown in this section in terms of flow patterns in the headpond and the sediment sluice channel, along with bed shear stresses through the headpond area.

Scenario 1: Design Flow Over the Coanda Screen (no sluicing)

Figure 6 shows the streamlines for the headpond and through the IFR gate for Scenario 1. The velocity magnitude is indicated by the streamline colour. The flow approaches the Coanda weir uniformly from the upstream end of the headpond. Velocities are low throughout the headpond (<0.5 m/s), with velocities of approximately 1 m/s over the Coanda screen and 1.5 m/s through the IFR gate.

Low velocities in the headpond resulted in low bed shear stresses (<1 Pa) for the entire headpond (shear stresses not shown for Scenario 1). A grain size of approximately 1 mm or smaller would be subject to incipient motion under these low shear stress conditions. Higher bed shear stresses (approximately 10 Pa) were predicted near the inlet to the IFR gate, which could result in incipient motion of a 10 mm grain size. The modelled bed shear stresses for Scenario 1 indicate that under normal operating conditions the headpond would be a zone of sediment deposition.

Figure 7 shows the streamlines for the headpond and near the sediment sluice gate for Scenario 2. The results indicate that the flow approaches the Coanda weir uniformly through most of the headpond with flow converging upstream and through the partially open sluice gate. The average velocity through most of the headpond is approximately

0.5 m/s or less. There is notable acceleration of flow towards the sluice gate with increased velocities starting at about 10 m upstream and reaching the maximum velocity of 8 m/s through the center of the sluice gate opening (velocity scale on Figure 7 shown to 2 m/s only to achieve better resolution throughout the headpond).

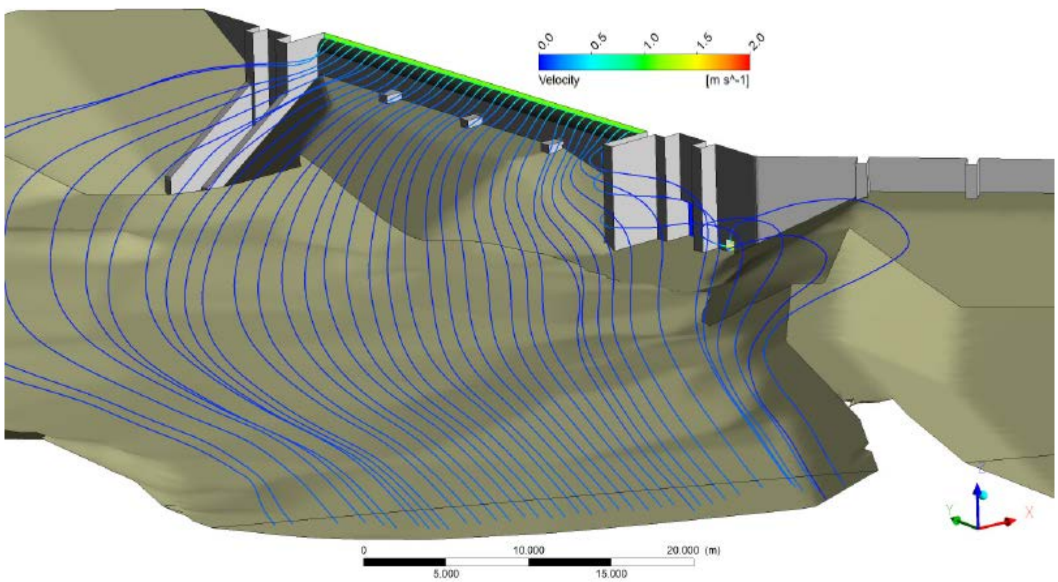

Figure 6: Streamlines through the headpond for Scenario 1

Scenario 2: Flow Over the Coanda Screen and through Sediment Sluice Gate

Figure 8 shows the modelled bed shear stress for the entire headpond. The low velocities in the headpond result in low bed shear stresses (<1 Pa) for most of the headpond, while the accelerating flow immediately upstream of the partially open sediment sluice gate results in increased bed shear stresses (up to 300 Pa). The lower bed shear stresses in the headpond would result in incipient motion of 1 mm or smaller grain sizes. The zone of bed shear stresses that are higher than 1 Pa is relatively small with a radius of about 7 m from the sluice gate opening. The shear stresses increase further from 10 Pa to 70 Pa within a radius of 2 m, and grain sizes of 10 mm to 70 mm could potentially be mobilized in this area. Right through the sluice gate opening, the velocities and the bed shear stresses are very high (8 m/s and 300 Pa, respectively), and as a result a grain size of 300 mm could be potentially mobilized (shear stress scale on Figure 8 shown to 10 Pa only to achieve better resolution throughout the headpond). The modelled bed shear stresses for Scenario 2 indicate that most of the headpond would be a zone of sediment deposition for sediment sizes larger than 1 mm, with some sediment mobility near the sediment sluice gate.

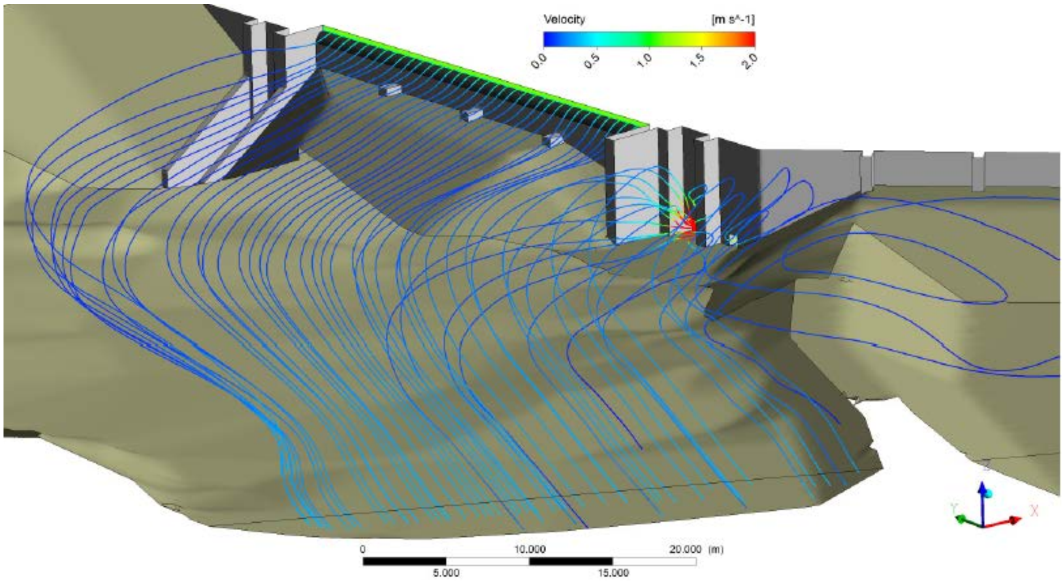

Figure 7: Streamlines through the headpond for Scenario 2

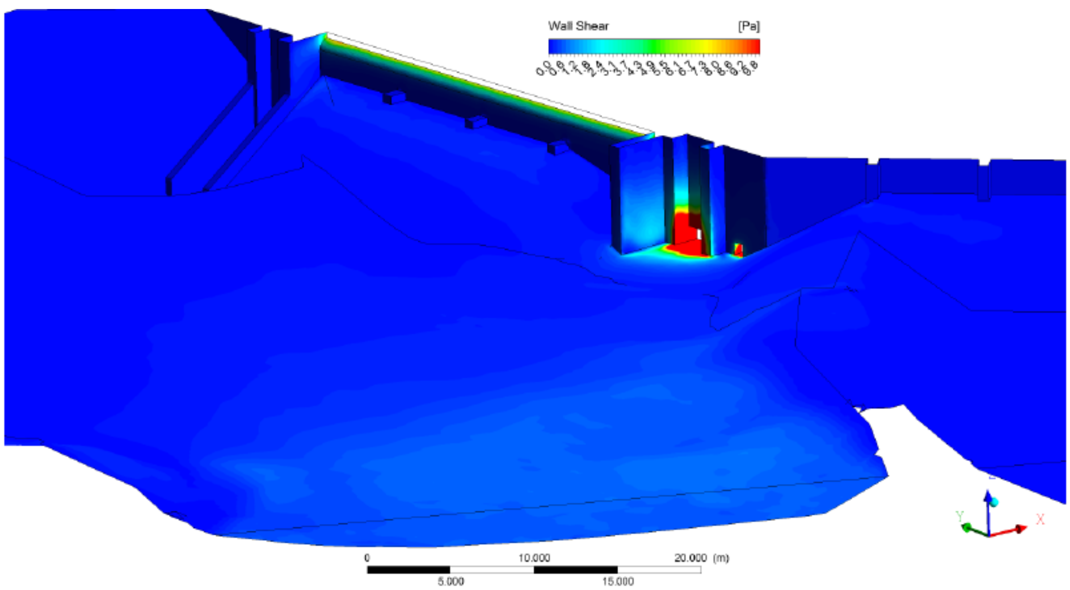

Figure 8: Shear stresses through the headpond for Scenario 2

Scenario 3: Flow through Sediment Sluice Gate Fully Open

Figure 9 shows the streamlines for the headpond and sediment sluice gate for Scenario 3, when all of the flow is passed through the fully open sluice gate and no flow passing over the Coanda Screen. This Scenario results in a lower water surface elevation in the headpond than in Scenarios 1 and 2. Higher velocities ranging between 1.0 and 2.0 m/s are observed along the headpond thalweg. A backwater effect is observed upstream of the sediment sluice gate that extends for about 30 m, which is consistent with the HEC-RAS model result for this Scenario.

Figure 10 shows the modelled bed shear stresses and associated incipient motion grain sizes for the entire headpond. Bed shear stresses are the highest along the headpond thalweg and are higher at the upstream end of the headpond due to smaller depths and higher velocities in this region. The grain sizes that are likely to be mobilized in the headpond at distances 40 m to 70 m upstream of the sluice gate are between 30 mm and 60 mm. The bed shear stresses are reduced within the lower velocity region caused by the backwatering up to approximately 30 m upstream of the sluice gate. The grain sizes that are likely to be mobilized through the backwatered region are 10 mm to 25 mm. Velocities and shear stresses increase again immediately upstream of the sluice gate as the flow accelerates through the sluice gate opening. In the radius of approximately 2 m from the sluice gate opening, the shear stresses increase above 100 Pa, and in this region grain sizes of about 100 mm can potentially be mobilized.

CFD Modelling Summary

The CFD model results provide a preliminary overview of the expected sediment mobility in the headpond based on initial headpond morphology. It is likely that during normal operations (Scenarios 1 and 2), sediment entrained from the upstream channel will deposit within the headpond and will result in most of the headpond becoming gradually infilled. Through time, the velocities and bed shear stresses are expected to deviate from those shown on Figures 6 through 10, depending on the modified headpond morphology. Additional studies of infilled headpond morphologies due to normal operations would enhance the understanding of the effect of sediment sluicing and sediment mobility through the sluice channel through time. Operating the sluice gate at various opening levels with different flow rates would also result in different upstream water levels, velocities and shear stresses, which would in turn impact the size of sediment mobilized in various regions of the headpond. Additional modelling of these alternate scenarios would further enhance our understanding of sluicing operations and associated sediment mobility.

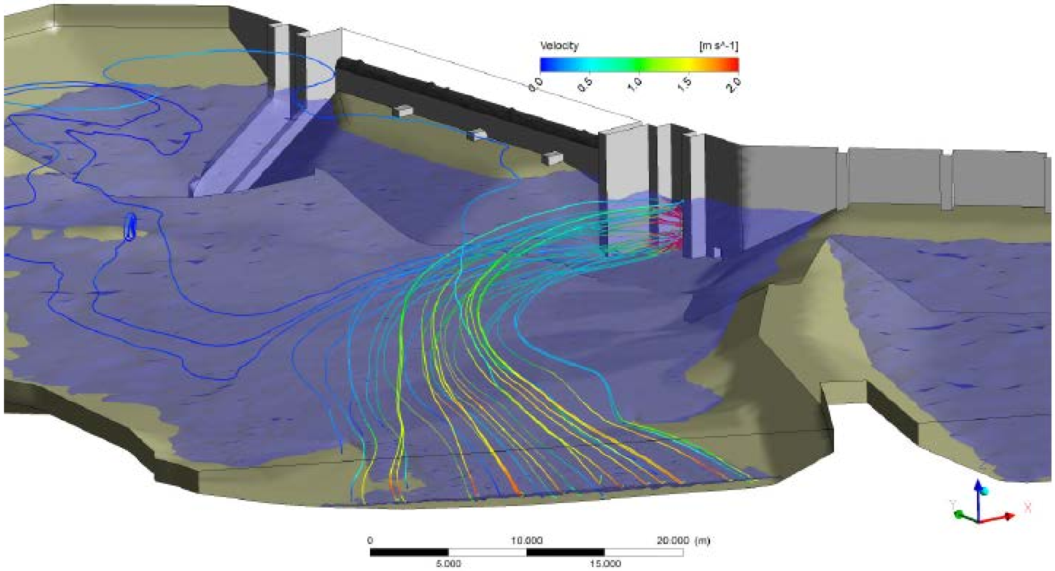

Figure 9: Streamlines through the headpond for Scenario 3

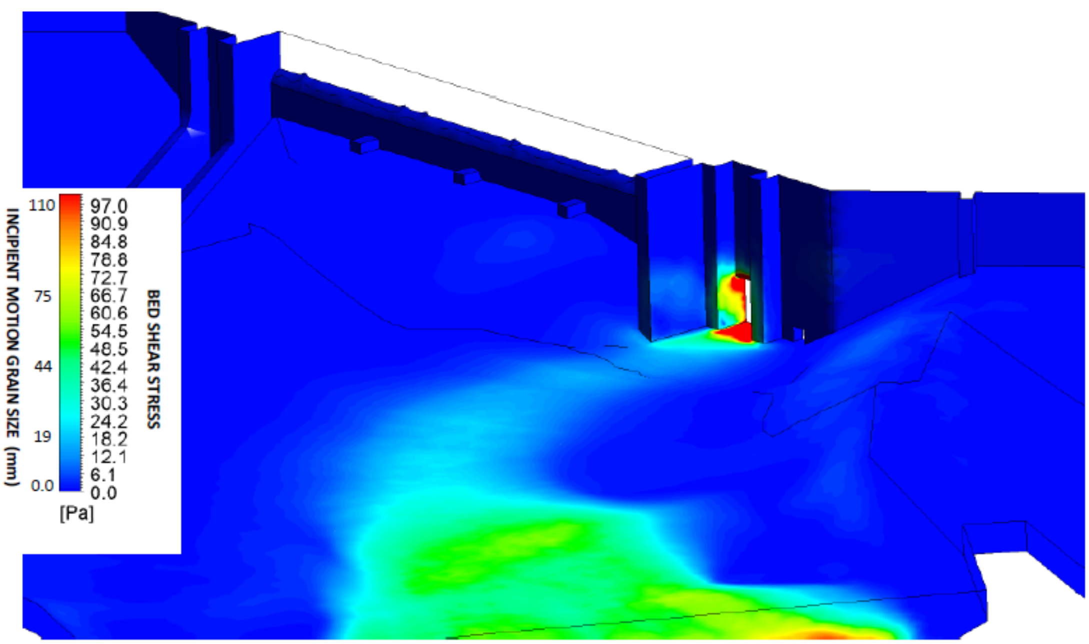

Figure 10: Shear stresses through the headpond for Scenario 3

4.3 Two-Dimensional Model

The FLO-2D model was used to investigate the sediment mobility in the natural stream channel prior to the construction of the intake facility. A grain size of 50 mm was used in these model runs to represent the size that is considered appropriate for fish spawning, which ranges between 10 mm and 100 mm based on field observations.

Figure 11 and Figure 12 show the results of the FLO-2D model and indicate areas of sediment deposition and scour at the end of 12 m^3/s and 24 m^3/s simulation runs, respectively (note that different scales for deposition and erosion are used on these two figures). While there are large areas of deposition in both scenarios particularly upstream and through the smaller side channel around an island, there are also areas within the single thread stream channel where a sediment size of 50 mm would be eroding. A comparison of Figure 11 and Figure 12 shows that deposition and erosion occur in similar locations, but that the predicted magnitude of scour is considerably higher for the higher stream flow. The full range of bed elevation changes for Scenario 4 is -0.5 m (erosion) to +0.5 m (deposition), while the range for Scenario 5 is -1.0 m (erosion) to + 1.0 m (deposition). Typical erosion depth for Scenario 4 is 0.15 to 0.20 m, while for Scenario 5 it is 0.25 to 0.50 m.

Further analysis could be conducted to determine the range of particle sizes that have a potential to be mobilized at 12 m^3/s and 24 m^3/s, since the modelling undertaken in this analysis is based on the 50 mm particle size only.

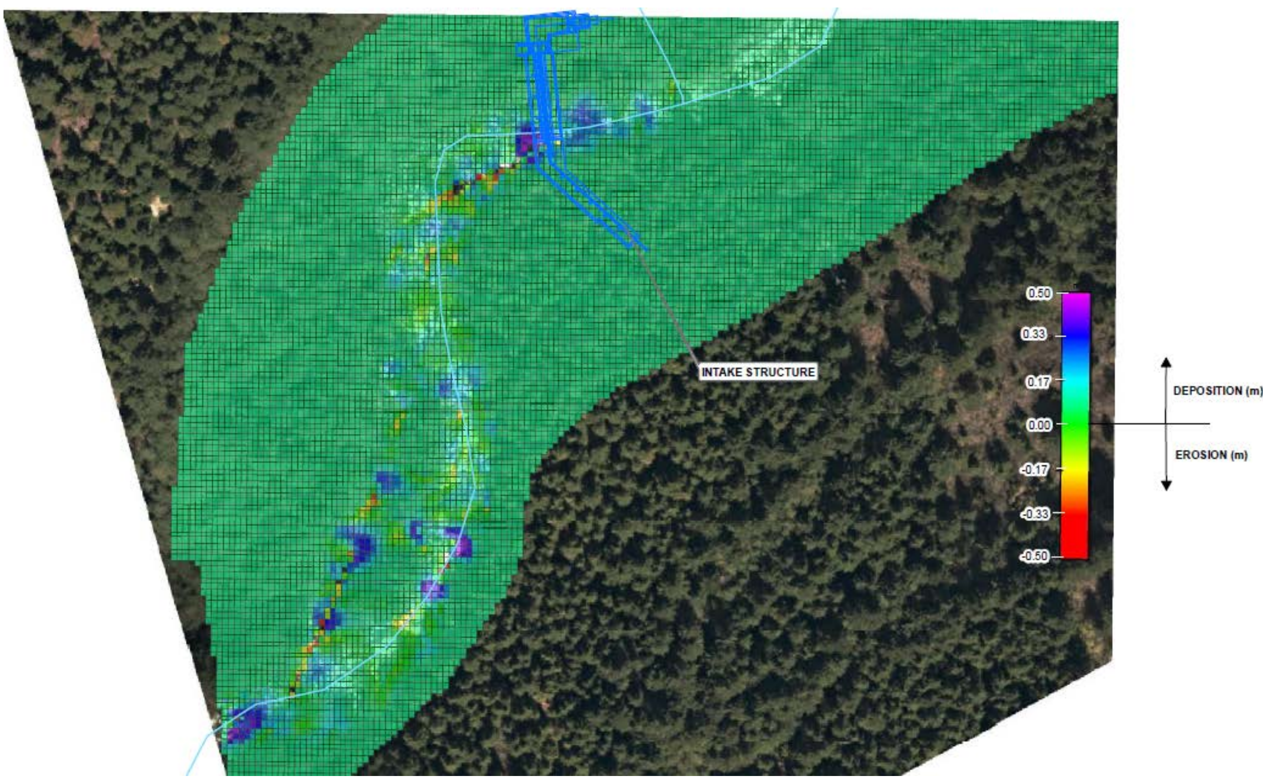

Figure 11: Baseline erosion and deposition of a 50 mm grain size for Scenario 4

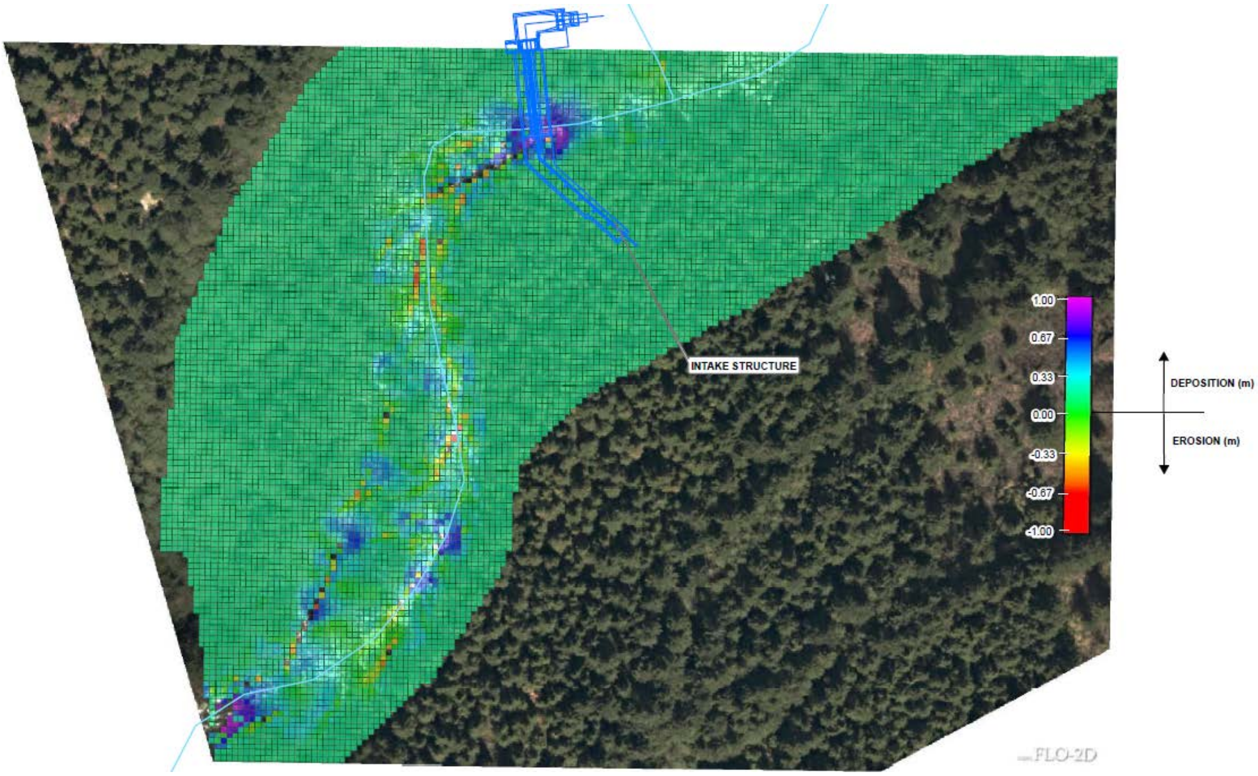

Figure 12: Baseline erosion and deposition of a 50 mm grain size for Scenario 5

5. Discussion and Conclusions

Flow patterns, depths and velocities within the headpond and at the intake structure during various flow scenarios were assessed using 1D (HEC-RAS) and CFD (ANSYS FLUENT) models. The 1D model was used to determine the large scale hydraulics and to provide boundary conditions for the CFD model, while the CFD model was used to examine flow patterns, velocities, shear stresses, and sediment incipient motion. A 2D (FLO-2D) model was used to examine sediment transport potential under baseline conditions in the natural channel upstream and downstream of the proposed intake location.

CFD modelling results for normal operations without sluicing activities (Scenario 1) indicate that the flow would approach the Coanda screen uniformly with low velocities throughout the headpond and maximum velocities over the Coanda screen and through the IFR gate. The low bed shear stresses simulated for this scenario indicate that the headpond is expected to be a zone of sediment deposition. Increased bed shear stresses that are localized near the IFR gate have a potential to mobilize an approximate grain size of 10 mm.

Similar to Scenario 1, CFD modelling results for sediment sluicing activities during normal operations (Scenario 2) indicate that flow would approach the Coanda screen uniformly with part of the flow converging through the sluice gate opening. Velocities are predicted to be low throughout the headpond and would start to accelerate approximately 10 m upstream of the sluice gate. The simulated bed shear stresses indicate that during this flow scenario most of the headpond would be a zone of sediment deposition and that sluicing activities with the headpond full would likely not result in a major passage of sediment to downstream reaches. There is a limited potential to mobilize sediment sizes ranging from approximately 10 mm to 70 mm, but only within a 2 m radius of the sluice gate opening.

If the headpond was drawn down during sluicing at high flow events (Scenario 3), the sediment mobility through the sediment sluice channel would greatly improve. The 1D and CFD modelling indicates that backwatering would still occur upstream of the sluice gate as the narrow gate opening forms a restriction to flow. The backwater would extend for about 30 m creating a zone of lower velocities and shear stresses. The simulated shear stresses indicate that during this flow scenario there would be a zone of increased shear stresses through the headpond thalweg (old stream channel) with up to 60 mm grain sizes being potentially mobilized.

In general, the sluice channel is expected to facilitate sediment transport through the headpond during flushing events (Scenario 3). It has been shown that there is a region of backwatering during these events that still creates a zone of lower velocities, which limits the size of sediment that would move in that area. It is expected that the bed slope and the velocities would change and enhance the predicted sediment transport as the headpond infills during normal operations, or as the sluicing activities progress and the sediment starts moving into these slower areas. The completed model simulations are for initial conditions only and additional modelling would be required to verify that sediment mobilization would occur after the headpond morphology was altered during normal operations or sluicing activities.

The baseline sediment mobility for the natural stream was examined using a 2D model (FLO-2D). The average monthly freshet flow in June (Scenario 4) and a reasonability high annual daily flow event (Scenario 5) were modelled to evaluate the potential sediment scour and deposition for a 50 mm grain size that occurs upstream and downstream of the proposed intake structure. Both simulations found that 50 mm sediment would have a potential to be mobilized in the reach upstream of the intake during these higher flow events, while the area right at the proposed intake appears to be a zone of sediment deposition in its existing natural state. The higher flow event resulted in more scour through the channel, indicating higher mobility of the 50 mm grain size.

Based on this study and after comparing the pre- and post-project conditions, it appears that the sediment sluice channel planned for the project would be effective in passing of the spawning size gravel that is similar to existing conditions in the natural channel.

References:

ANSYS (2011). *ANSYS Fluent Version 14.0 – User's Guide and Theory Manual*. ANSYS, Inc. Canonsburg, PA, USA.

Buffington, J.M., and D.R. Montgomery (1997). A Systematic Analysis of Eight Decades of Incipient Motion Studies, with Special Reference to Gravel-Bedded Rivers. Water Resources Research, 33, 1993-2029.

Church M., M.A. Hasan, and J.F. Wolcott (1998). Stabilizing Self-Organized Structures in Gravel-Bed Stream Channels: Field and Experimental Observations. Water Resources Research, 34, 3169-3179.

Einstein, H.A. (1968). Deposition of suspended particles in a gravel bed. Amer. Soc. Civ. Eng., Vol. 94, No. HY5, pg.1197-1205, Sept. 1968.

FLO-2D Software Inc. (2012), FLO-2D Reference Manual, Nutrioso, Arizona, www.flo-2d.com.

Martin V. (2003). Hydraulic Roughness of Armoured Gravel Beds: the Role of Grain Protrusion. Ph.D. Thesis, Department of Civil Engineering, University of British Columbia, Canada.

Meyer-Peter, E., and R. Müller (1948). Formulas for bedload transport, *2nd Meet. Int. Assoc. Hydraul. Structures Res.,* Stockholm, (Appendix 2), 39-64.

Shields, A. (1936). Anwendung der Äenlichkeits-Mechanik und der Turbulenzforschung auf die Geschiebebewegung, *Mitteilungen der Preussische Versuchsanstalt für Wasserbau und Schiffbau*., Berlin, Heft 26. Translation by W. P. Ott and J. C. van Uchelen, Soil Conservation Service, California Institute of Technology, Pasadena.

Zeller, M. E. and W. T. Fullerton (1983). A Theoretically Derived Sediment Transport Equation for Sand-Bed Channels in Arid Regions. Proceedings of the D. B. Simons Symposium on Erosion and Sedimentation, R. M. Li and P. F. Lagasse, eds., Colorado State Universily and ASCE.