ASSESSING SEDIMENT MOVEMENT BY CFD PARTICLE TRACKING

Jose (Pepe) Vasquez, Hydraulic Engineer, Northwest Hydraulic Consultants, 30 Gostick Place, North Vancouver, BC, 604-980-6011 (Ext. 211), jvasquez@nhc-van.com

Abstract The potential application of the three-dimensional (3D) computational fluid dynamics (CFD) model Flow-3D to simulate the movement of non-cohesive sediment particles transported passively by the water flow through hydraulic structures, was assessed. Three tests cases: (1) a desander; (2) a run-of-river intake and (3) a powerhouse intake in a reservoir were used. In each case, the sediment paths computed by Flow-3D appeared correct. For example, the deviation of near-bed sediment from the main flow direction caused by secondary currents in curved flows was reproduced. However, the model cannot accurately reproduce the movement bed load sediment, because the interaction with the bed is not accounted for in the model. Nevertheless, the CFD particle tracking technique seems promising for simulating suspended sediment movement through complex hydraulic structures.

INTRODUCTION

Conventional morphodynamic models are designed to compute bed changes in alluvial rivers caused by sediment transport. These models can compute bed erosion or deposition at fixed cells or elements within a defined computational domain. Although such models have proved useful in many river engineering applications, they become problematic to apply in cases involving hydraulic structures exposed to sediment laden flows, such as water intakes in rivers.

One of the main challenges of water intake structures built in natural alluvial rivers is to prevent the excessive ingestion of sediment carried by the diverted water. For example, since the early work of Bulle (1926), it is well known that secondary currents can cause a disproportionate amount of sediment to be diverted into lateral intakes. This is because the slow-moving water near the channel bed has less inertia and turns easily into the intake, while the faster moving surface water continues moving down the river. Since most sediment travels near the bed, a large portion of the total sediment load in the river can be ingested by the intake, even if the diverted water flow is relatively small. To minimize this problem, intakes are normally located above the bed level or protected by sills, weirs or bottom-current deflectors. As a result, the flow around the intake is usually highly three-dimensional (3D). Conventional sediment transport models have problems dealing with such complex flows, leaving in many cases physical modeling as the only practical tool for the design of some of these structures.

An alternative numerical approach is investigated here by using a 3D computational fluid dynamics (CFD) model with Lagrangian particle tracking capabilities. The CFD model can use fixed cells (Eulerian frame of reference) to compute the flow field on top of which Lagrangian (moving) particles can be superimposed. It is here hypothesized that this approach could be use to simulate the passive movement of coarse non-cohesive sediment transported by the water flow. The advantage of a CFD model is that complex 3D structures and flows can be readily simulated.

To illustrate the potential of this technique, it was applied to three test cases. In the first qualitative test case, the settling of sand within a desander was verified. In the second qualitative test case, the movement of sediment into a run-of-river intake was tested. Finally, the amount of sediment passing into a diversion tunnel and powerhouse measured in a physical model was compared with the results of the CFD particle tracking technique. In all cases the bed remained fixed (i.e. bed level changes were ignored). It was found that bedload sediment transport is difficult to simulate correctly because the interaction with the bed is not properly accounted for in the model. Also, it was found that the results could be sensitive to the initial conditions of the particles if they sediment source is close to the study area. Despite these limitations, CFD particle tracking seems promising for practical engineering applications, especially when dealing with suspended sediment at complex hydraulic structures. These preliminary results encourage further research on this technique.

FLOW-3D PARTICLE TRACKING

The commercial CFD model Flow-3D was used in all tests. Flow-3D allows the release of particles from a user-defined source. A rectangular block can be defined as a source of particles in any part of the computational grid, with a given generation rate expressed in particles/second. The particles are passively transported by the flow; they do not interact with each other or with the fluid (unless specified otherwise). The particles are assumed as spherical and defined by properties such as diameter and density. For simulating natural sediment a density of 2650 kg/m^3 was adopted for all tests. The path of the particles can be tracked and the number of particles crossing a given surface can be documented. In this way, it may be possible to estimate, for example, the proportion of sediment entering into an intake. However, the particles lack volume and hence the model cannot be used to predict bed changes caused by sediment deposition.

SIMPLE DESANDER

The first qualitative test was performed in a simple rectangular desander to test the logic of this approach. A narrow channel 2 m wide by 2 m deep expands abruptly into a wider desander channel 4 wide by 4 m deep. The average flow velocity reduces from 1.0 m/s in the approach channel to 0.25 m/s in the desander, promoting suspended sediment settling along the 50 m length of the desander. Particles were released at the inflow section to simulate sand grains depositing within the desander. The particles tracks for D = 0.1 mm and D = 0.2 mm can be seen in Figure 1. Based on the settling velocity of each particle, the theoretical length L needed for a particle to settle in a 4 m deep desander flowing at 0.25 m/ is L = 120 m for the D = 0.1 mm and L = 40 m for the D = 0.2 mm; so, it is expected that most of the 0.2 mm particles will settle within the desander. This behavior seems to be qualitatively reproduced by the model (Figure 1).

RUN-OF-RIVER INTAKE

The second application was performed for a large run-or-river intake located in a steep mountain river, where coarse sediment up 70 mm has been observed to be lifted by the flow during floods. Since the intake is located along the inside of a sharp bend, it is expected that secondary currents would divert large amounts of sediment into the intake (Bulle 1926). Figure 2 shows the general layout of the structure and a screenshot of particles between 1 and 10 mm. About 90% of the incoming flow comes upstream from the 90-degree bend, forcing the flow to make a sharp turn into the intake. As expected, a large portion of the incoming particles are diverted into the Sluiceway Channel, some of them overcome a high sill at the Forebay Control Structure and make their way into Intake Forebay and from them into the Power Tunnel.

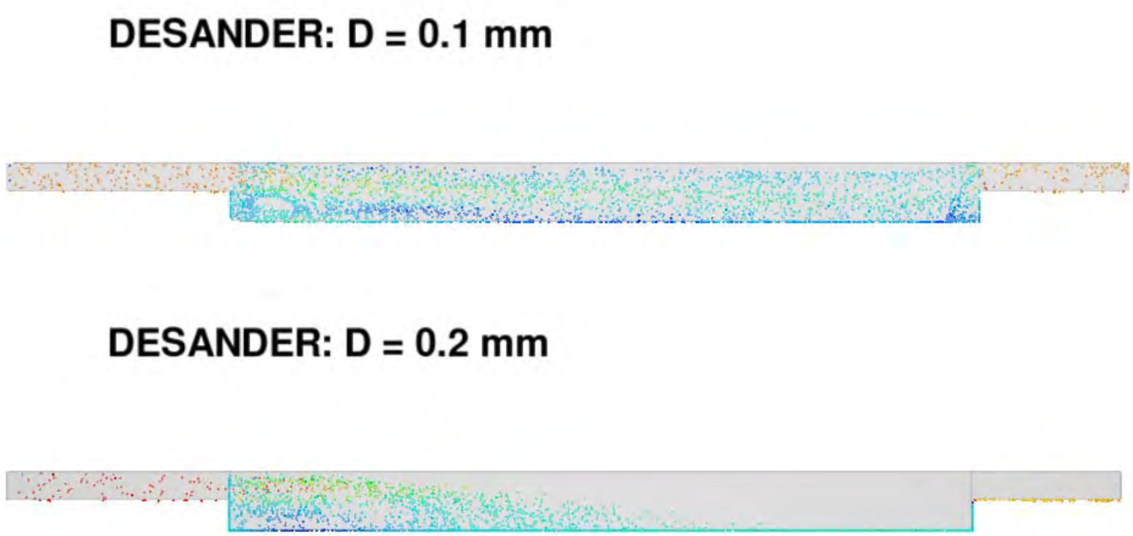

Figure 1 Comparison of particles tracks inside a desander for 0.1 mm and 0.2 mm “sand” particles.

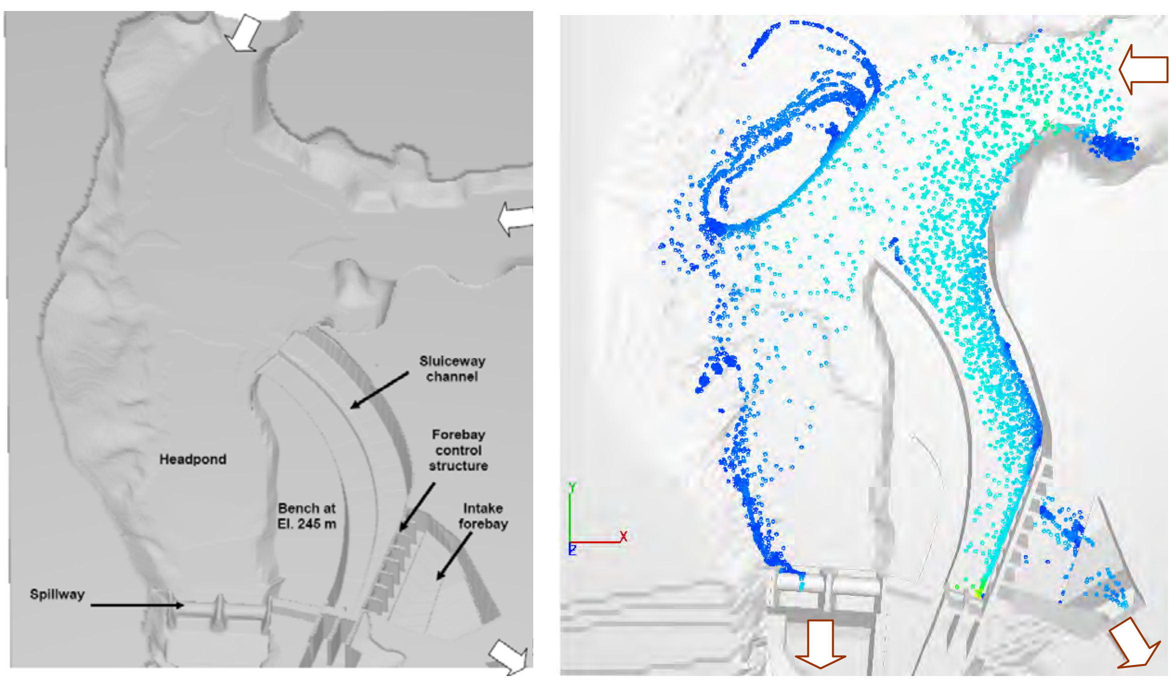

Figure 2 Run-of-river layout and computed particles tracks.

POWERHOUSE INTAKE IN RESERVOIR

The third application consisted of evaluating flow through the powerhouse and spillway located in a reservoir covered by coarse 1 mm sand (Figure 3). Flow velocities within the reservoir are high enough to move the sand as bedload, but not in suspension. The powerhouse contains four units, each with an independent water intake. The unit located farthest upstream – Unit 4 - was subject to intense sediment ingestion. From results of a physical model study it was decided to build a sediment diversion tunnel upstream from Unit 4. Measurements of the percent of sediment passing through the powerhouse and tunnel made in the physical model, for the conditions before and after the introduction of the tunnel, are shown in Table 1.

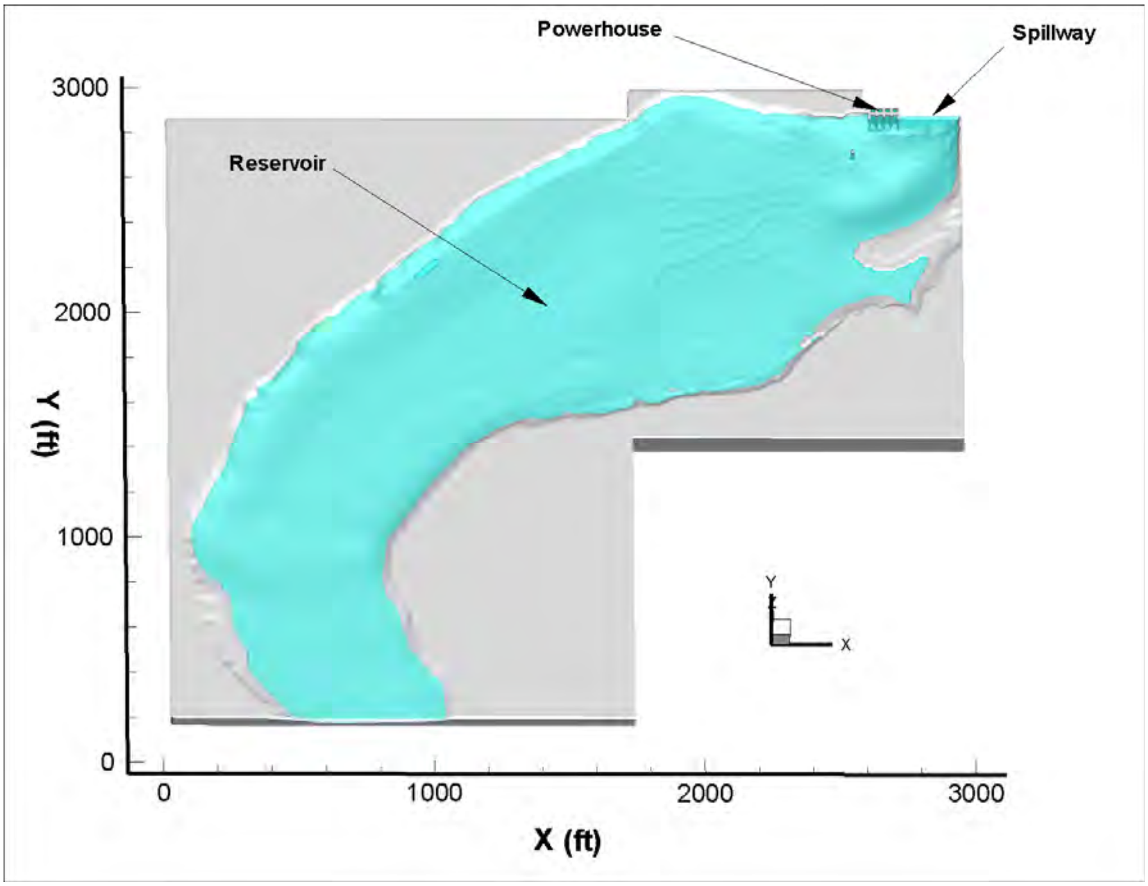

Figure 3 Modeling domain showing reservoir, powerhouse and spillway.

Table 1 Sediment passage measurements in the physical model.

	Pre-tunnel	Post-tunnel
Tunnel	-	96%
Unit 4	94%	2%
Unit 3	3%	2%
Unit 2	1%	0%
Unit 1	1%	0%

Once the Flow-3D model was calibrated by reproducing the velocity patterns observed in the physical model, particles representing 1 mm sand were released in the reservoir, 300 ft upstream from the powerhouse from a sediment source 200 ft wide. Figure 4 shows the particle tracks for the pre-tunnel and post-tunnel conditions, and Table 2 shows the results of the CFD model.

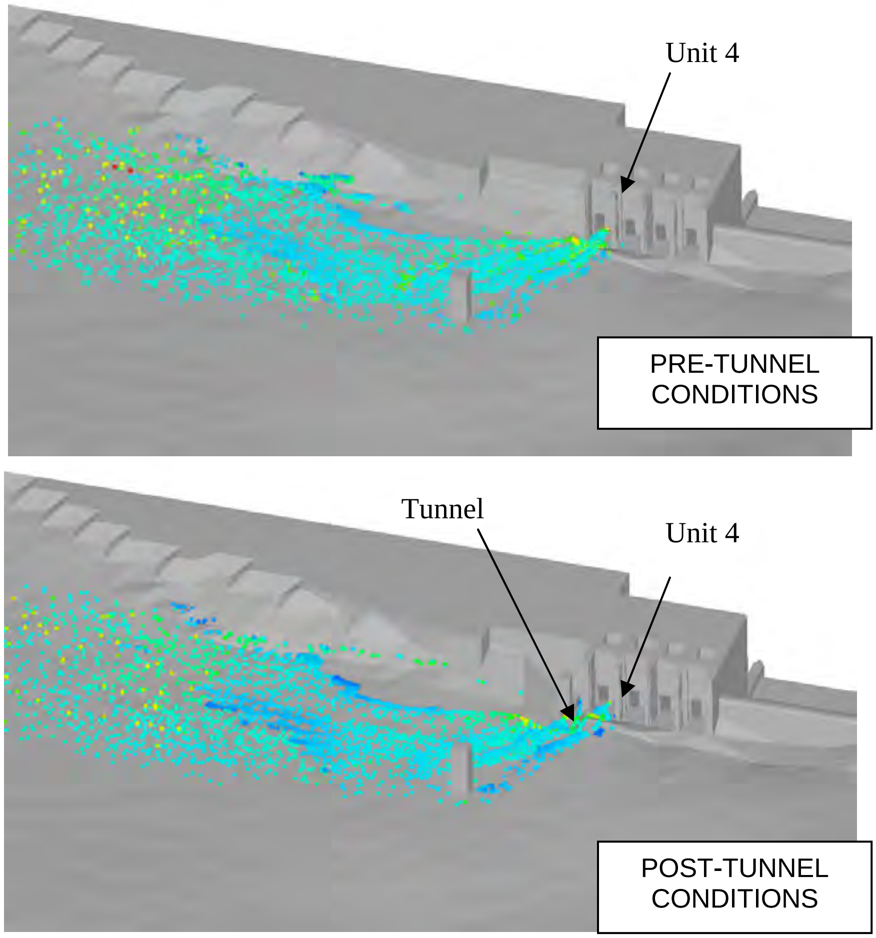

Figure 4 Particle track simulations for the pre-tunnel and baseline conditions.

Figure 4 shows that upstream from the powerhouse particles move parallel to the left bank, but as they approach the tunnel and powerhouse they quickly turn almost 90 degrees into the tunnel and Unit 4. This behavior seemed to be in general agreement with the results of the physical model, suggesting that the CFD model is capturing the main physics of the particle movement; however, the agreement between the physical model diversion rates (Table 1) and the CFD model diversion rates (Table 2) was only qualitatively correct.

Furthermore, the CFD results as presented were achieved only after numerous iterations were made, included changing the size and resolution of the mesh, varying the location and size of the sediment source, changing the downstream water level and smoothing the bed topography. The remaining discrepancy between the physical and numerical model results and the sensitivity of the numerical model results to adjustments in the mesh and sediment source characteristics make the reliability of the numerical model results somewhat questionable (even recognizing the uncertainty related to the physical model results).

Table 2 Results of CFD model.

	Pre-tunnel	Post-tunnel
Tunnel	-	78%
Unit 4	100%	22%
Unit 3	0%	0%
Unit 2	0%	0%
Unit 1	0%	0%

DISCUSSION OF RESULTS

The CFD particle tracking model was found to capture important physical features of sediment movement; such as the effects of sediment size (Figure 1) and the effects of secondary currents on deflecting sediment paths (Figures 2 and 4). Therefore, the results seem qualitatively correct and probably provide fairly reliable results for applications involving suspended load dominated flows. However, simulations involving bed load transport were found to be problematic and quite sensitive to the set-up of the model, as discussed below.

Bedload Transport Perhaps the main limitation of the CFD particle tracking technique is associated with modeling bedload transport, or the fraction of sediment moving in contact with the bed (as opposed to suspended in the water column). The movement of sediment over a riverbed is influenced not only by the flow dynamics (bottom shear stress) and sediment size, but also by the bed topography (local bed slope, depressions, mounts), intergrain friction, imbrication and particle interaction - none of which are properly accounted for by the CFD model. Bed depressions (holes) and adverse bed slopes were found to be quite problematic, as sediment tended to be captured in these bed features. Under natural conditions, sediment will fill in a depression, and once full incoming sediment will continue travelling downstream passing over the infilled depression. However, since the model particles lack volume, the depression will never be filled and it becomes a virtual particle sink.

Particle Source Block The location and size of the sediment source block was also found to have a strong influence on the amount of particles reaching and passing through the powerhouse, tunnel and spillway in the last case analyzed. For example, Figure 5 shows how a narrow source block located a short distance upstream from the powerhouse resulted in most of the particles exiting through the tunnel; while a wider source block at the same location increased the volume of particles exiting through units 3 and 4.

The particle source block adopted for the model calibration seemed to produce reasonable agreement with the physical model results (Table 2 and Figure 4). However, its calibration by trial-and-error was possible only because physical model data was available. Attempts to locate the sediment source farther upstream failed because sediment moved near the bed and was trapped in intermediate bed depressions before reaching the powerhouse.

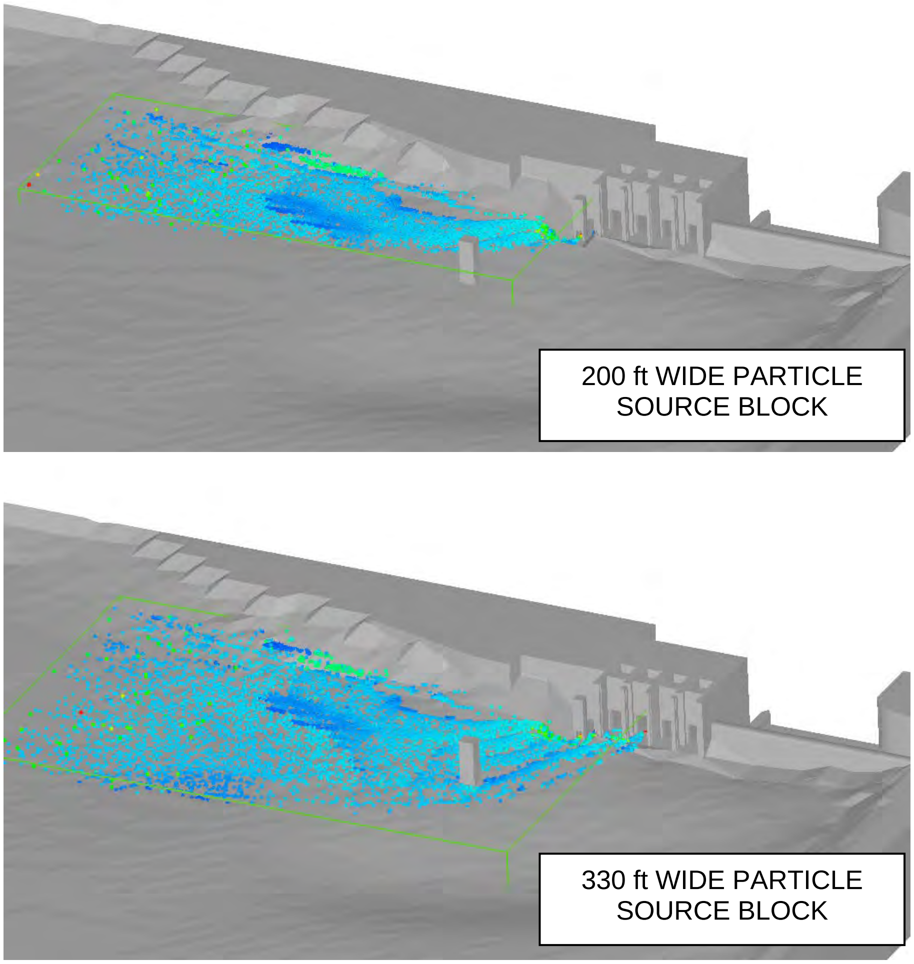

Figure 5 Effect of the source block width on particle tracks.

Suspended load Transport The CFD particle tracking technique is probably more appropriate for simulating the movement of sediment by suspension or saltation, when the particles do not interact strongly –or at all - with the bed. However, since the interaction between the particles is ignored, the effect of sediment concentration is also ignored.

It can be concluded that despite of the apparent limitations of the CFD particle tracking approach, it appears to be a very promising technique to study the sediment movement of suspended sediment in complex 3D hydraulic structures; as a complement, or perhaps an alternative, to physical modeling.

REFERENCES

Bulle, H. (1926). Untersuchungen uber die Geschiebeableitung bei der Spaltung von Wasserlaufen. VDI Verlag, Berlin (in German).