Prepared in cooperation with the Federal Interagency Sedimentation Project (FISP)

Computational Fluid Dynamics Analysis of Suspended-Sediment Sampler Efficiency

Justin A. Boldt[1] and David S. Mueller[2]

[1] Hydrologist, U.S. Geological Survey, Kentucky Water Science Center, Louisville, KY, USA, jboldt@usgs.gov
[2] Hydrologist, U.S. Geological Survey, Office of Surface Water, Louisville, KY, USA, dmueller@usgs.gov

Physical Modeling, 1941

Background

The design of suspended-sediment samplers, like the one shown in Fig. 1., is based on detailed laboratory flume tests conducted in 1941 (Fig. 2). These experiments documented the effect of the relative sampling rate (or intake efficiency) on the sampled concentration (Fig. 3). The intake efficiency is defined as the ratio of the nozzle velocity to the ambient stream velocity. The majority of the effort in the design of existing and new samplers is spent in trying to achieve isokinetic performance of the sampler over a wide range of stream conditions (depth, velocity, temperature, and transit rate). The key variables of interest are the intake efficiency, sediment size, and stream velocity.

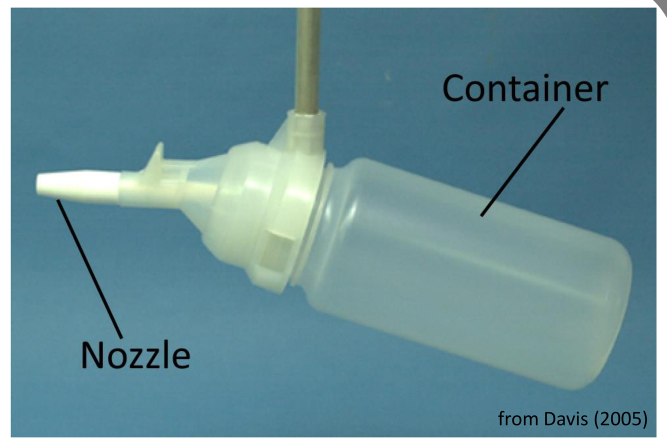

Figure 1. US DH-81 suspended-sediment sampler.

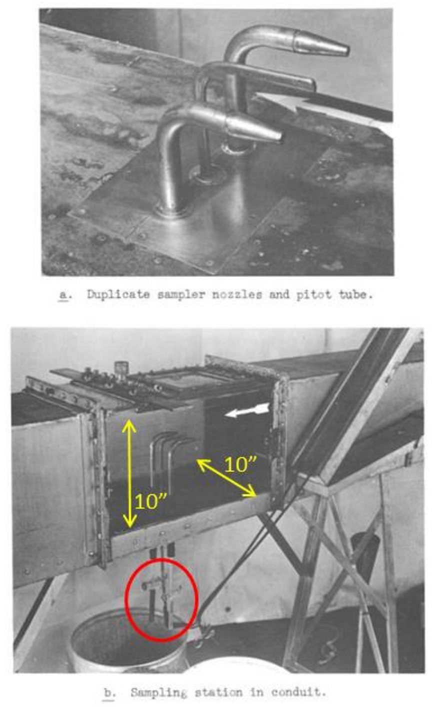

Figure 2. Sampling nozzles and flume from the 1941 lab study. Sampling intake rate was controlled by the values (circled in red).

Research objective:

Evaluate and verify the 1941 lab results with numerical modeling.

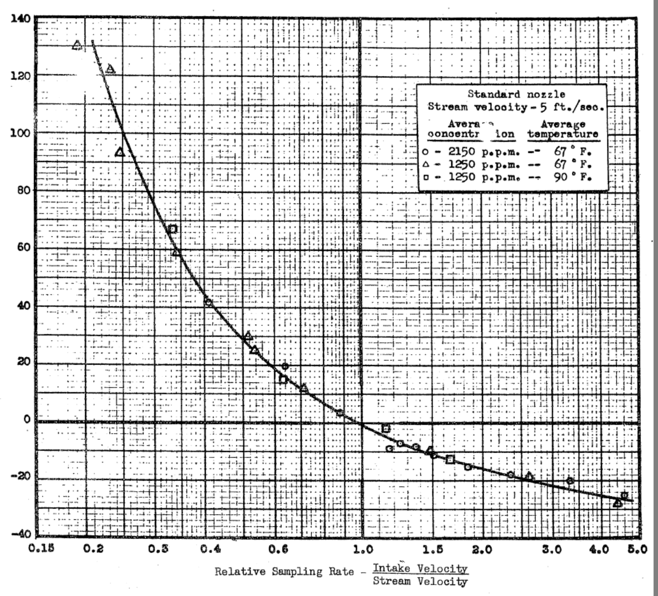

Figure 3. Effect of relative sampling rate on sediment concentration (FISP, 1941).

Abstract

The measurement and characterization of sediment transported by streams is of vital importance to the effective management of water resources. Since its formation in 1939, the Federal Interagency Sedimentation Project (FISP) has worked to develop standard and scientifically valid equipment and methods for collecting sediment samples. All of the FISP suspended-sediment samplers have the same general operating principle—a water-sediment sampler is collected isokinetically through a nozzle and into a container (Fig. 1). The foundational design assumption for isokinetic samplers is that the water velocity at the intake nozzle must match the ambient stream velocity; otherwise, a bias will be introduced into the concentration of the collected sample. The effect of non-isokinetic sampling conditions on the sampled concentration has previously been researched with laboratory flume tests and field tests. An alternative to physical testing is to use computation fluid dynamics (CFD) modeling. **In this study, the use of CFD modeling with particle tracking capability allows a detailed evaluation of the effect of intake efficiency on the sampled sediment concentration in simulated turbulent flow.** The results of this research include a better understanding of the hydrodynamic characteristics that are important to isokinetic sampling, an independent analysis of isokinetic sampling and sample concentration bias for variable sediment sizes, and a basis to adjust current design criteria for existing samplers to ensure that they are being manufactured to be both accurate and cost-effective. This study was conducted in cooperation with the Federal Interagency Sedimentation Project (FISP).

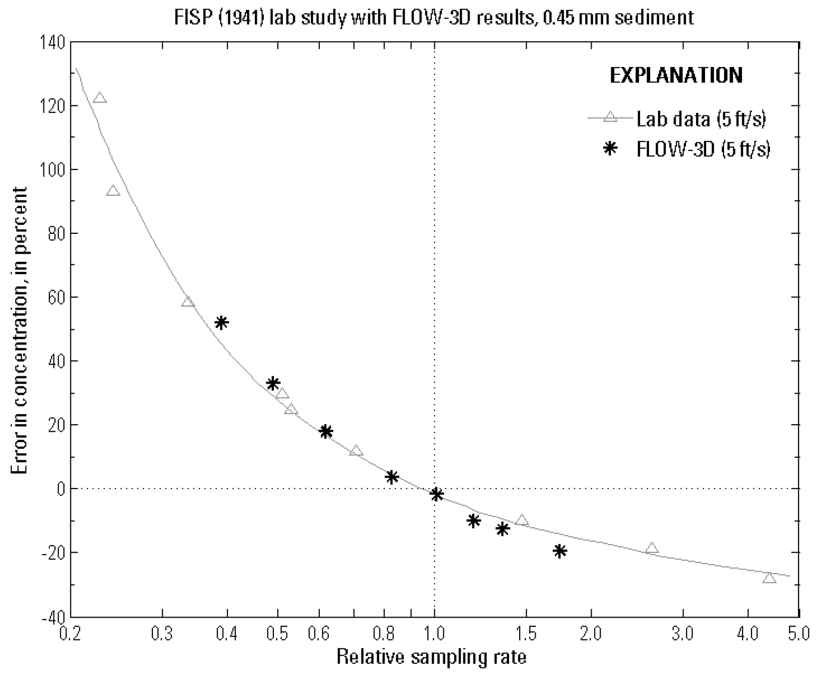

Figure 9. FLOW-3D results compared to FISP (1941) lab study, 0.45 mm sediment, 5 ft/s.

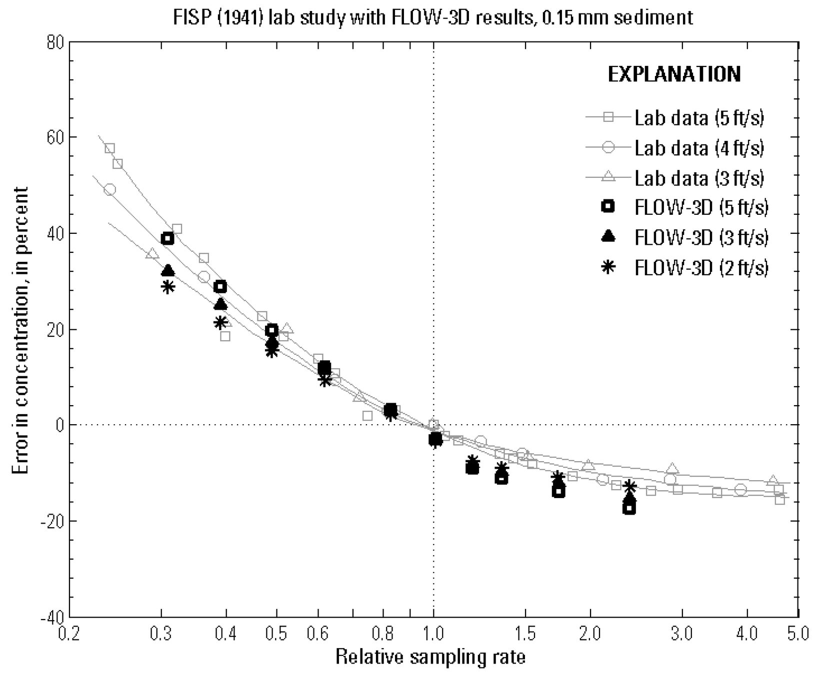

Figure 10. FLOW-3D results compared to FISP (1941) lab study, 0.15 mm sediment, 2–5 ft/s.

Visualization of Streamlines and Particles

A major benefit of numerical modeling is the ability to visualize streamlines, particles, or any other fluid dynamics variables of interest.

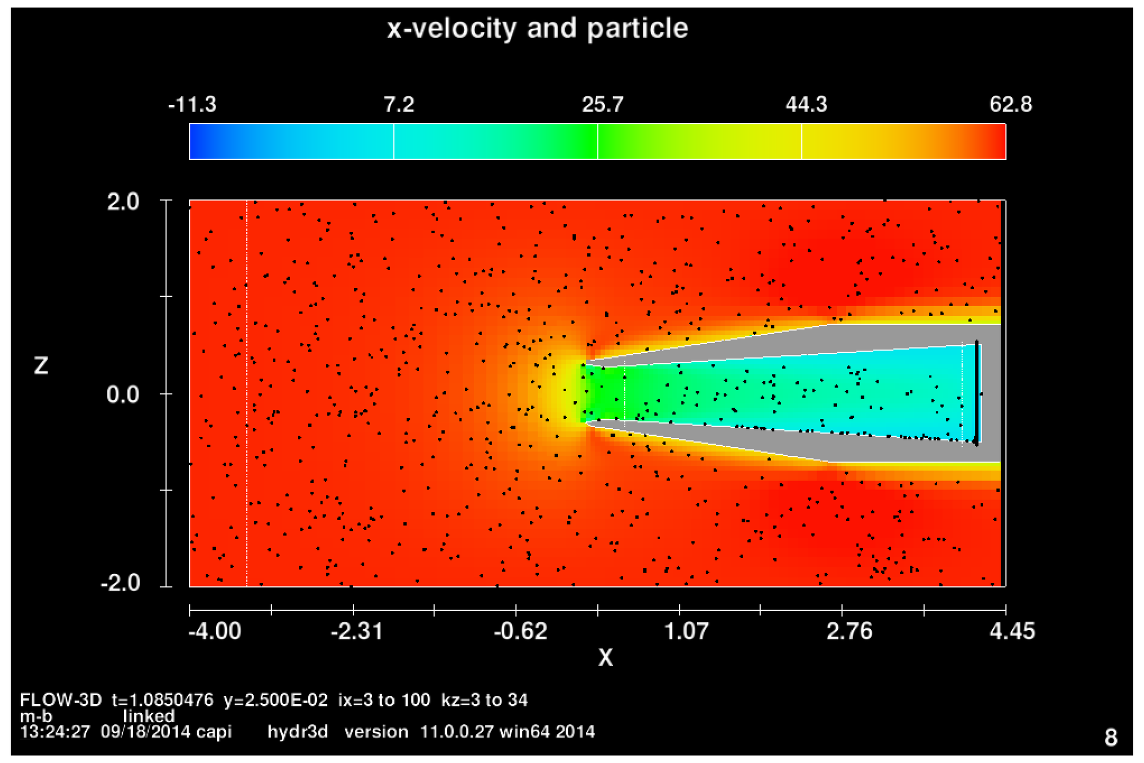

Figure 11. Contour plot of x-velocity with 0.15 mm particles.

The model uses clear water conditions (water temperature held constant at 20°C) with Lagrangian particle tracking. Mass particles with a specific gravity of 2.65 were introduced into the flow at a continuous source regularized in space and time (uniform distribution) at the upstream boundary. The particles are spherical with a uniform density and uniform density and a constant particle size distribution (Fig. 11). Two different particle sizes were tested—0.45 mm and 0.15 mm. The particle rate was specified such that the resulting ambient SSC was comparable to the 1941 lab conditions (~1250 mg/L). A two-way (fully coupled) particle/fluid momentum transfer model was activated (i.e., full fluid-particle interaction). The particle motion is influenced by the fluid flow through the drag forces.

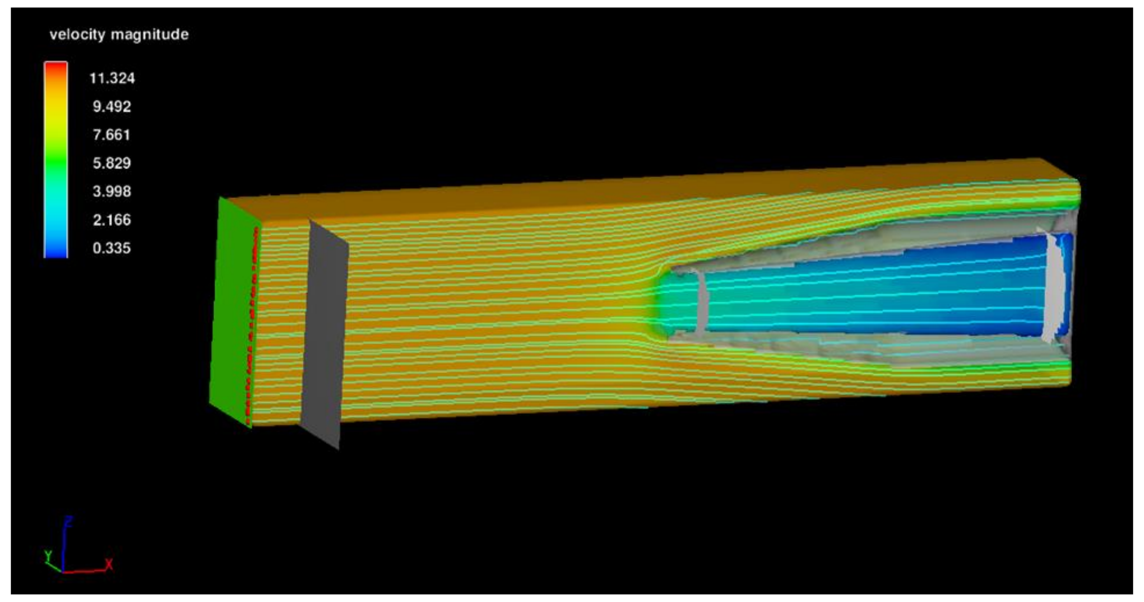

Figure 12. Streamlines around the nozzle for an intake efficiency less than 1.

Numerical Modeling, 2014

Methods

The numerical simulations were performed with FLOW-3D version 11.0 (Flow Science, Inc., 2014). FLOW-3D is a computational fluid dynamics (CFD) software package with multi-physics modules. It solves the three-dimensional Navier-Stokes and continuity equations in a structured, rectangular grid. Each simulation was performed using twelve cores on a Dell Workstation with two Xeon 3.33 GHz processors (6 cores/processor) and 12 GB RAM running Windows Server 2012.

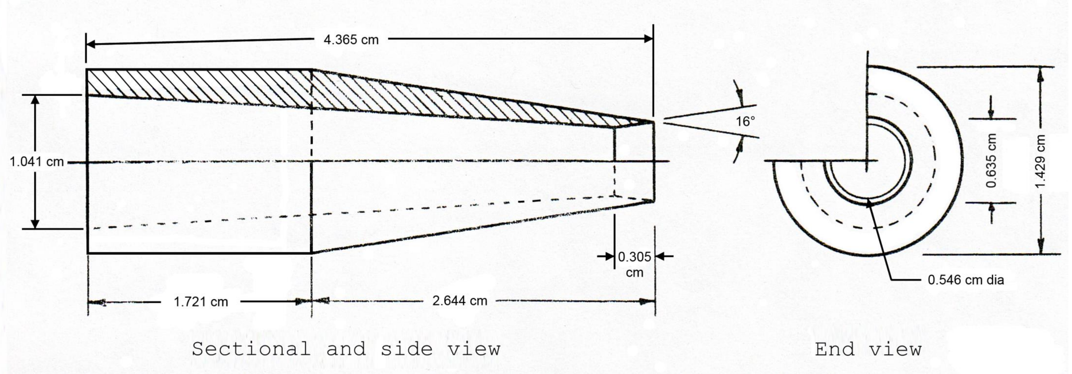

Figure 4. Technical drawing of the nozzle (FISP, 1941).

The nozzle geometry (Fig. 4 and 5) is embedded into the computational grid (Fig. 6) by the FLOW-3D pre-processor using the Fractional Area-Volume Obstacle Representation (FAVOR) technique.

The back end of the nozzle was blocked off with a cap. A sink was placed at the very back of the nozzle on the inside and just in front of the cap (Fig. 7). The sink is used in this model for two reasons. The first is to be able to control the intake rate in order to vary the nozzle intake efficiency. This is analogous to the lab study where the fluid was controlled by a valve and tube attached to the nozzle. The second is to remove fluid within the domain at a defined volume flow rate. The removal of fluid represents the passing of fluid through the nozzle and into a sample bottle.

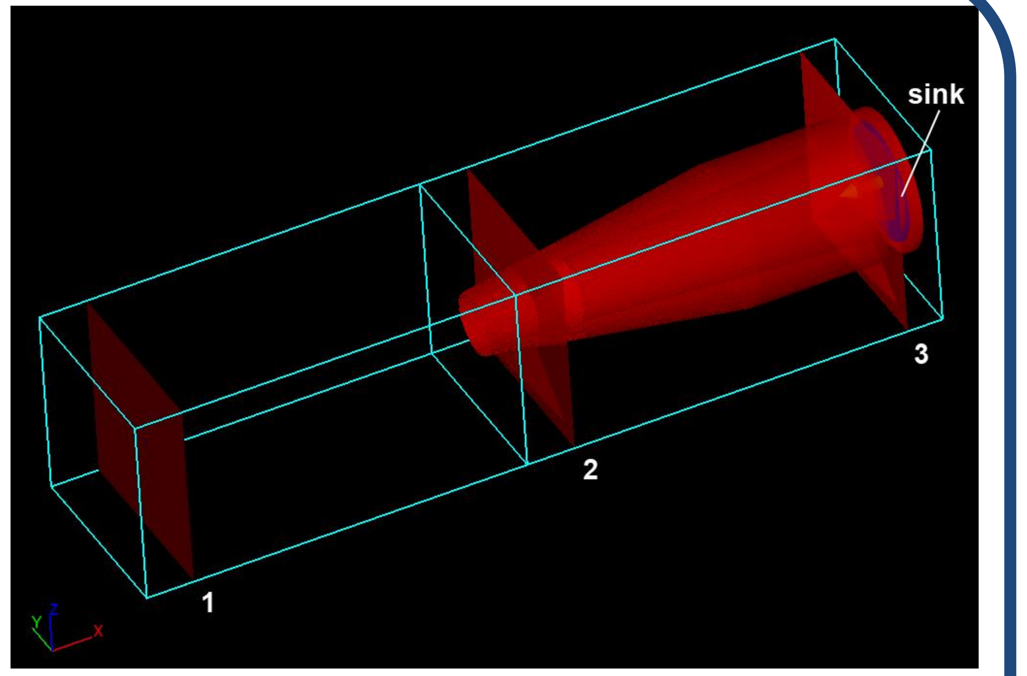

Figure 7. Nozzle geometry setup in FLOW-3D.

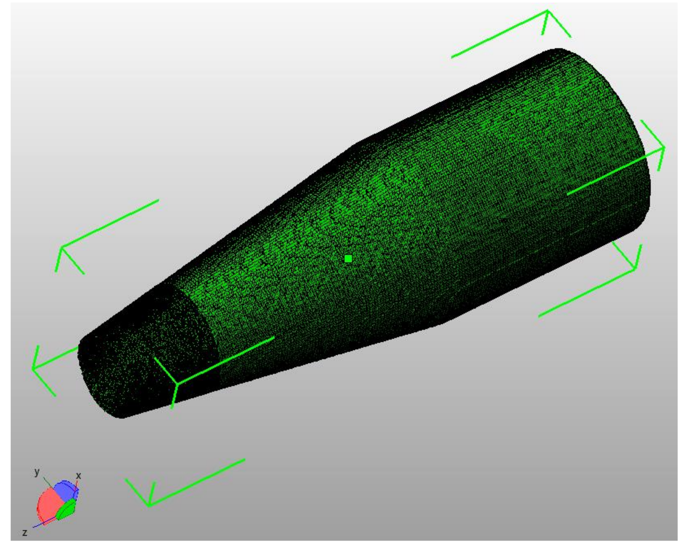

Figure 5. A computer-aided drawing (CAD) package was used to create a 3-D rendering of the nozzle.

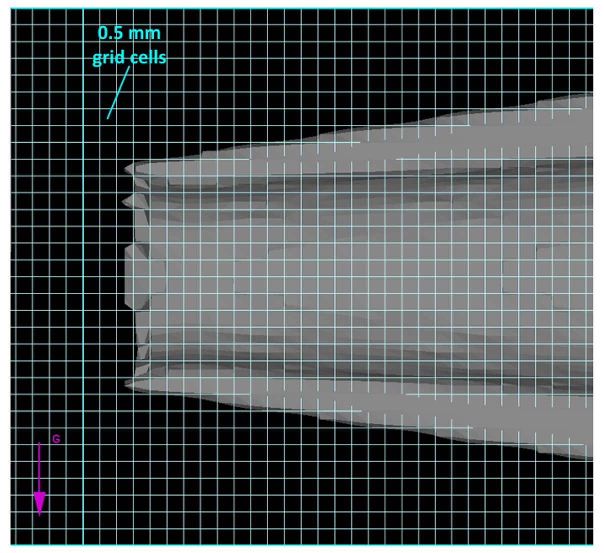

Figure 6. Nozzle geometry within FLOW-3D with 0.5 mm grid cells.

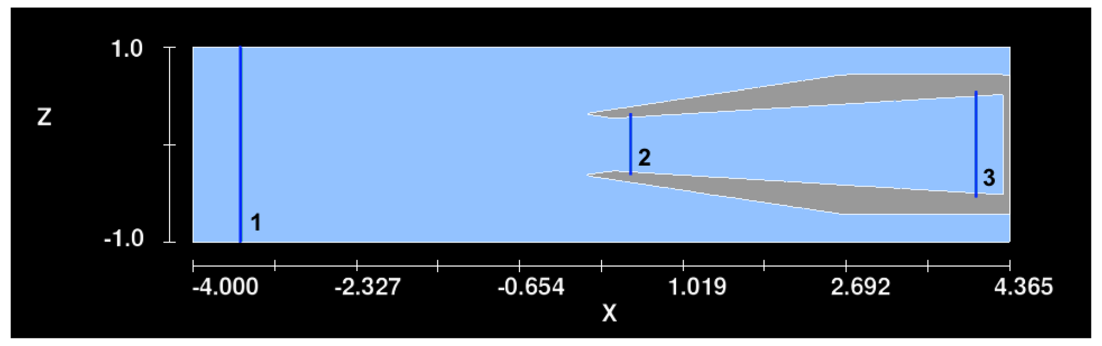

Figure 8. FLOW-3D nozzle geometry with flux planes.

Flux planes (Fig. 8) were used to measure fluid volume flow rate, flux surface area wetted by fluid, and particle counts (total number of particles crossing the flux surface) at three locations. Nozzle velocity and suspended-sediment concentration (SSC) can be calculated from these measurements and are then compared to the ambient conditions to determine the error.

Results

The simulation results from the FLOW-3D model compare very well with the 1941 lab data (Fig. 9 and 10). The maximum error is less than 5%, which is excellent for a sediment sample. The deviations from the lab data are likely a result of the particle physics. The sediment used in the lab study was not completely uniform in size and shape.

The results indicate that the bias error in concentration is greater for larger particle sizes (Fig. 9) and at faster stream velocities (Fig. 10). This relates to the theory of drag force, which is a function of an object's area and relative velocity.

When the relative sampling rate (intake efficiency) is less than 1, which means the nozzle velocity is less than the stream velocity, the streamlines bend around the nozzle (Fig. 12). Non-isokinetic sampling causes a bias in the sediment concentration because of changes in the streamlines and the difference in density of the water and the sediment. Sediment particles are denser than water and respond more slowly to curves in the streamlines, which causes an excess of particles in the sample for intake efficiencies less than 1, and vice versa.

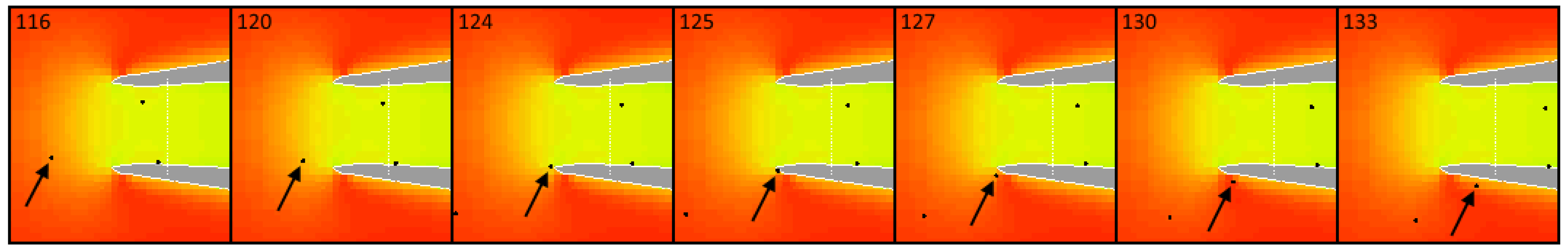

Figure 13. Frames from a video showing a particle bouncing off the tip of the nozzle.

Summary and Conclusions

- The 1941 lab results were validated with numerical modeling using the FLOW-3D CFD model.
- This research provides the foundation for additional use of CFD to evaluate and develop design specifications for existing and future nozzles and samplers.
- Future work includes testing different nozzle designs, mixed sediment sizes, and natural fill conditions.

References

Davis, B.E. (2005). Report QQ: A guide to the proper selection and use of federally approved sediment and water-quality samplers. Federal Interagency Sedimentation Project.

Federal Interagency Sedimentation Project (1941). Laboratory investigation of suspended sediment samplers, Report 5, 99 p.

Flow Science, Inc. (2014). FLOW-3D v. 11 user's manual, Flow Science, Inc., Santa Fe, NM.