Authors:

Timothy Sassaman
Engineer II
Alden Research Laboratory, Inc.
30 Shrewsbury St, Holden, MA 01520-1843
Phone (508) 829-6000 ext. 6435

Andrew Johansson
Director of Hydraulic Modeling
Alden Research Laboratory, Inc.
30 Shrewsbury St, Holden, MA 01520-1843
Phone (508) 829-6000 ext. 6435

Ryan Jones
CFD Engineer II
Alden Research Laboratory, Inc.
30 Shrewsbury St, Holden, MA 01520-1843
Phone (508) 829-6000 ext. 6435

Marianne Walter
Civil Engineer
Consumers Energy | Engineering Services Department
1945 W Parnall, Jackson MI 49201
Phone (517) 788-0050

HYDRAULIC ANALYSIS OF A PUMPED STORAGE POND USING COMPLEMENTARY METHODS

1.0 ABSTRACT

The Ludington Pumped Storage Plant is located along the shores of Lake Michigan in Ludington, Michigan. It was constructed between 1969 and 1973, and is jointly owned and operated by Consumers Energy and DTE Energy. At the time of construction, it was the largest pumped storage hydroelectric facility in the world. The plant consists of an upper reservoir that is connected to Lake Michigan (lower reservoir) by penstocks equipped with six reversible pump-turbines with a total designed power capacity of 1,872 MW. When the price of electricity is low the motor driven pumps are used to fill the upper reservoir with 27 billion gallons (82,300 acre ft) of water; which is referred to as pumping operation. In the second phase of the cycle, generating operation, water is released from the upper reservoir and flows through the turbines thereby generating power when the price of electricity is high.

Due to an anticipated increase in the need for on demand (dispatchable) power to offset fluctuations in green energy power production (wind and solar power) and maintenance needs, there was a desire to upgrade the plants' existing pump-turbines and generators to provide more dispatchable power. The upgrade also has the benefit of increasing the life span of the plant.

In 1969 the upper reservoir was modeled at Alden Research Laboratory to evaluate vortex formation at the intake for generating operations, and potential scour of the reservoir lining during pumping operations. The original study was used to develop a vortex suppression baffle wall and a reservoir bed protection plan.

To evaluate the impacts of the upgrade project on the upper reservoir, a hydraulic study combining multiple techniques was conducted. The study included field measurements, a physical hydraulic model, and a computational fluid dynamics (CFD) model of the site.

Field velocity data was collected in the upper reservoir during pumping and was used to validate both the physical and numeric models. The CFD model was used to evaluate flow patterns and identify areas of potential increase in scour to be evaluated further in the physical model. The physical model was used to evaluate potential hydraulic issues related to both pumping and

generating operations in the upper ueservoir. For pumping operations, the physical model was used to identify potential areas of concern associated with scour, flow patterns and reservoir fill rates in the upper reservoir. In generation mode, the physical model provided insight into vortex formation at the intake structure, flow patterns and reservoir draw down rates in the upper reservoir.

This paper presents a comparison of the results from the two studies, and evaluates the complementary roles of the different study tasks. The paper also evaluates differences in the two modeling approaches, and outlines advancements made in hydraulic modeling between the time of the original study and the current study.

2.0 INTRODUCTION

2.1 Owner Background

Consumers Energy Company (CEC) is a combination electric and natural gas utility company that provides service to about 1.75 million electricity customers and about 1.6 million natural gas customers in all 68 counties of Michigan's Lower Peninsula. Overall, CEC provides natural gas and/or electricity to more than six million of Michigan's nearly 10 million residents. CEC's customer base includes a mix of residential, commercial, and diversified industrial customers, the largest segment of which is the automotive industry. CEC is the principal subsidiary of CMS Energy. CEC and CMS Energy are both headquartered in Jackson, Michigan.

The Detroit Edison Company was founded in 1903 and is one of the largest electric utilities in the United States. Detroit Edison, now DTE Energy (DTE), is a full service regionally-integrated energy and energy-technology corporation. DTE generates and distributes electricity to 2.1 million customers in southeastern Michigan. With an 11,080 megawatt (MW) system capacity, DTE uses coal, nuclear, natural gas, wind, solar, and pumped-storage hydroelectric to generate its electrical output. DTE is headquartered in Detroit, Michigan.

2.2 Plant Background

The Ludington Pumped Storage Plant is co-owned by CEC and DTE and located in Pere Marquette and Summit Townships of Mason County, on the eastern shore of Lake Michigan (which serves as the Plant's Lower Reservoir). It is the Owners' largest peaking facility with a nameplate rating of 1,872 MW, providing enough electricity to support more than 1.3 million residential customers. The six unit pumped-storage hydroelectric plant has been recognized as one of the state's engineering marvels. Upon commissioning in 1973, the Plant was recognized by the American Society of Civil Engineers (ASCE) with the Outstanding Civil Engineering Achievement Award as a result of national competition that year. The Plant has a cycle efficiency of approximately 71.5%.

The six 312 MW (nameplate rating) pump-turbine/motor-generator units at the Plant can cycle from zero to 1,872 MW of generation in about 25 minutes as each unit is brought on-line. In addition, if a blackout were to occur, in the eastern United States or across the U.S. due to a catastrophic event, the Plant's design makes it an important facility to provide emergency restart power for a number of base load power plants.

As part of the Plant's overall maintenance program, the Plant is undergoing a maintenance upgrade (Overhaul) on each of the six units. The upgrade consists of replacement of the pump-turbine runners and the motor/generator stators including the windings. Modern pump-turbine and motor-generator designs allow significantly more efficient machines to be installed in existing footprints of each unit, resulting in increased capacity (uprate) utilizing the existing gross head range of approximately 295 to 362ft. The power-generating enhancements proposed for the Ludington Pumped Storage Plant will add 300 MW of installed generating capacity and will increase the Plant's hydraulic capacity at the best efficiency point and at mid-range net head by 9,690 cubic feet per second (cfs). This represents a 14.5-percent increase over the current hydraulic capacity of 66,600 cfs. The increased capacity increases the ability to balance the electrical grid more effectively and the Overhaul will extend the useful life of the plant. The Overhauls are currently scheduled to begin in the fall of 2013 and to finish in mid-2019 (all six units will be overhauled, one-unit-per-year, using winter overhauls).

2.3 Upper Reservoir Description

The upper reservoir consists of a lined reservoir and an intake structure where six (6) penstocks terminate/originate. Each penstock is about 1,300ft long with varying diameter from 28.5ft (at the intake) to 24ft (at the powerhouse). The intake structure is approximately 240ft wide with a baffle wall and six (6) 28.5 ft square bays where the penstocks transition from square to circular. East of the intake structure, a concrete apron extends 290ft from the face of the intake structure. The apron consists of two retaining walls, two splitter walls, and an additional wall at Unit #1 that protects the reservoir liner during filling following outages involving low pond levels. The reservoir has a clay liner along the bottom and an asphalt liner from elevations 860 to 950ft and operates between water surface elevations of 875 to 942ft.

2.4 Study Objectives

The objective of this study was to evaluate the impacts of the increase in flow capacity in both generation and pumping mode. To accomplish this, CEC and DTE contracted with Alden Research Laboratory (Alden) to perform field measurements, a physical hydraulic model, and a Computational Fluid Dynamics (CFD) model of the upper reservoir. This paper presents a comparison of the results from the two studies, and evaluates the complementary roles of the different study tasks. The paper also evaluates differences in the two modeling approaches, and outlines advancements made in hydraulic modeling between the time of the original study and the current study.

3.0 1969 PHYSICAL MODEL STUDY

3.1 Model Similitude

Prior to construction in 1969, a 1:122 physical scale model of the Ludington Pumped Storage Project's upper reservoir was modeled at AldenAlden before construction to evaluate hydraulic conditions of the prototype reservoir. The model study used Froude similitude for correlating the model and prototype relationships. At the time the 1969 study was conducted, similitude requirements to limit scale effects in the physical model were not as fully studied and understood as they are today, especially in terms of vortex formation. The Reynolds and Weber numbers at the intake with water at a temperature of 50 degrees Fahrenheit in the model were 2×10^4 and 113, respectively. Based on current similitude standards minimum Reynolds and Weber numbers of 3×10^4 and 120 respectively are required to avoid potential scale effects. (Jain, Dagget and Keuligan) The scale effects of importance for this study could result in significant surface tension effects leading to the underestimation of vortex strength in the model.

3.2 Study Results

The 1969 model study was enlightening in terms of the project site hydraulics. While the data collection techniques and model similitude may not have been consistent with present Hydraulic Institute of Standards (HIS), information from the study was helpful with the 2011 study design.

The 1969 model showed that the potential for scour was greatest with a mid-pool water level while pumping, not at the lowest water level. The jet that is created by the pumps discharging into the upper reservoir creates two large eddies on either side of the jet as shown in Figure 1. As the upper reservoir continues to fill the jet re-entrains flow increasing the momentum and the near-bed velocities. After the upper reservoir is half full the increased depth decreases the velocities. Figure 2 shows a photo from the testing showing dye injected into the discharge for flow pattern mapping.

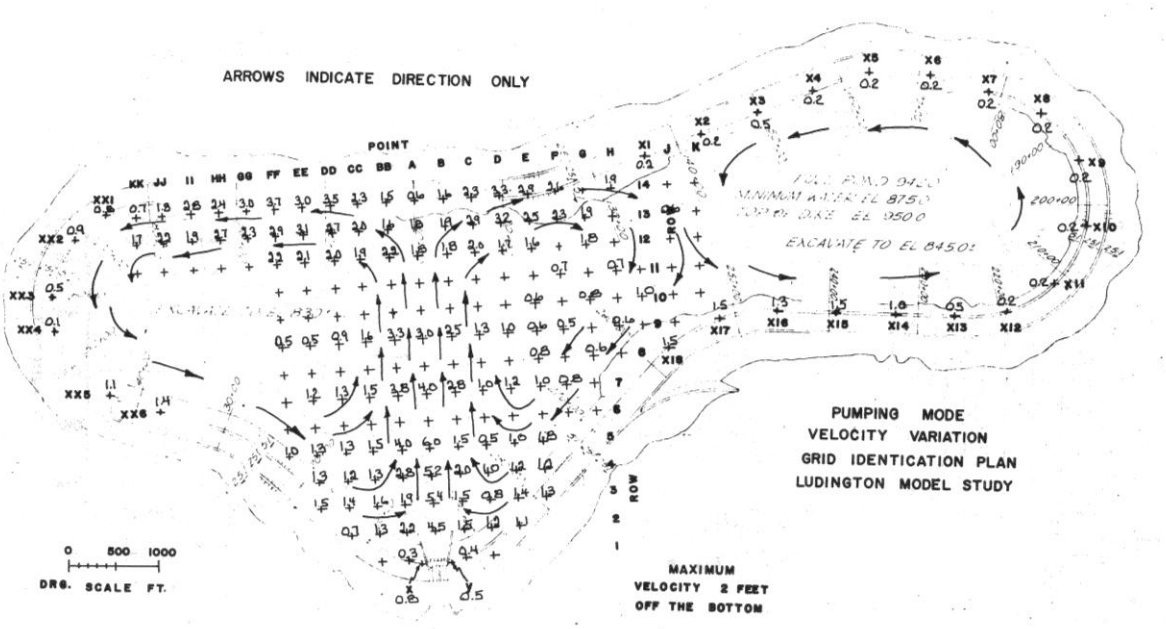

Figure 1 : Flow Patterns from the 1969 Model Study

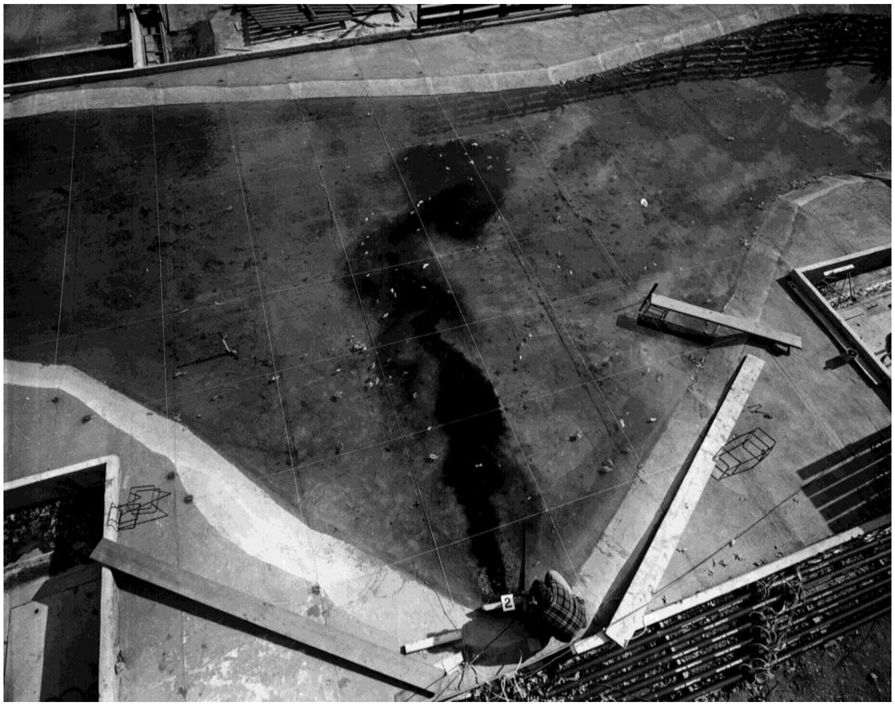

Figure 2 : Dye Injection in the Physical Model

3.3 Vortex Formation

Testing of the 1969 physical model showed vortex formation at the intake during generating. The classification used for vortex strength was air entraining vortices or ice entraining vortices. The HIS vortex classification system had not been developed when this physical model study was conducted. Based on the HIS vortex classification, the vortices that were observed in 1969 would correlate to Type 4 or stronger surface vortices. Figure 3 shows the HIS free surface vortex classification.

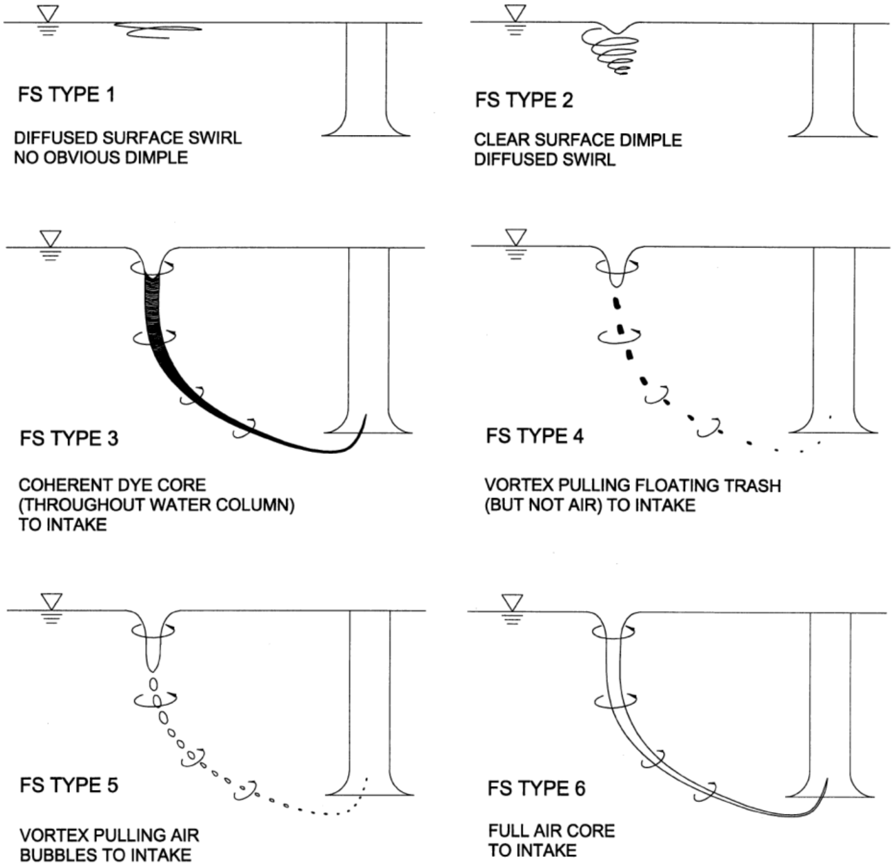

Figure 3 : Hydraulic Institute of Standards Free Surface Vortex Classification

A series of corrective measures including baffle walls and floating walls were developed and tested in the 1969 model until an acceptable design was reached. The final design developed from the model study results included a baffle wall that extended the full height of the intake structure down to the top of the intake passageway. The baffle wall had 72 four-foot diameter holes to allow some flow behind the wall to help dissipate/eliminate vortices. Figure 4 shows the baffle wall of the prototype structure.

Figure 4 : Prototype Intake Structure

3.4 Scour Potential

The physical model study was also used to develop a discharge apron design to limit the scour potential during normal reservoir filling operations. Prototype velocity measurements were made two ft above the reservoir bed using hot wire anemometers to record the velocity magnitude. The flow patterns were also mapped during filling and draining of the reservoir.

The model study tested different discharge apron configurations to minimize the near-bed velocities and also evaluated the use of dividing walls to more evenly distribute the flow exiting the discharge structure. Twelve different discharge apron designs were tested in the physical model until a design that limited the near-bed velocities to acceptable levels was found. The resulting design consisted of two dividing walls to separate the flow from the six pump-turbines into three jets, and an apron that rose from the discharge by about ten feet keeping the higher velocities towards the top of the water column. Figure 5 shows a photo of the final discharge apron that was developed for the study.

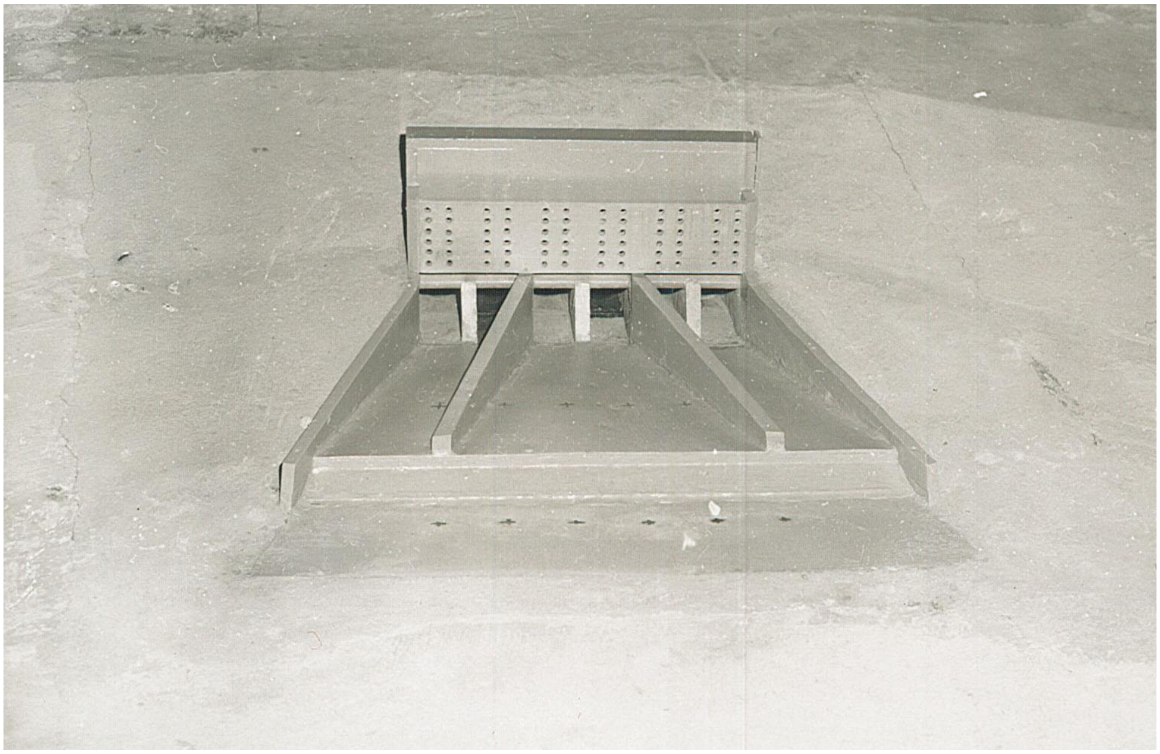

Figure 5 : Final Discharge Apron Design, 1969

The physical model study was also used to evaluate the potential for scour when the upper reservoir was first filled. The plan for filling the reservoir was to operate Unit 1 to fill the reservoir at a low rate. The main concern from filling the reservoir in this manner was that the initial fill could scour away the bed lining in front of Unit 1 due a hydraulic jump at the end of the discharge apron. A deflector wall was developed on the apron which forced the hydraulic jump to occur on the concrete apron, thus dissipating the energy from the filling of the reservoir before the bed liner would be worn away.

3.5 Limitations of the Physical Model Study

There were limitations in what could be learned from the 1969 physical model study. One main limitation was the potential for scale effects in the formation of vortices due to the low Reynolds and Weber numbers. These scale effects could cause the vortex strength to be under represented and possibly limit the number of vortices that could form.

4.0 2011 FIELD VELOCITY MEASUREMENTS

As part of the 2011 modeling effort, field velocity measurements were conducted in the upper reservoir. These measurements were made utilizing a boat mounted acoustic Doppler current profiler (ADCP) connected to a differential global positioning system (DGPS) positioning system.

A total of nine velocity transects were measured at two water levels while the upper reservoir was filling as shown in Figure 6. Velocity data were collected twice along each transect. A pair of full mappings was used to determine if any significant alterations to the flow patterns could be observed as the reservoir depth increased. The first mapping period was conducted as the reservoir filled to the nominal half-way point while the second mapping captured the velocity field as the reservoir completed filling. This data were used to verify the flow patterns in the new CFD and physical models.

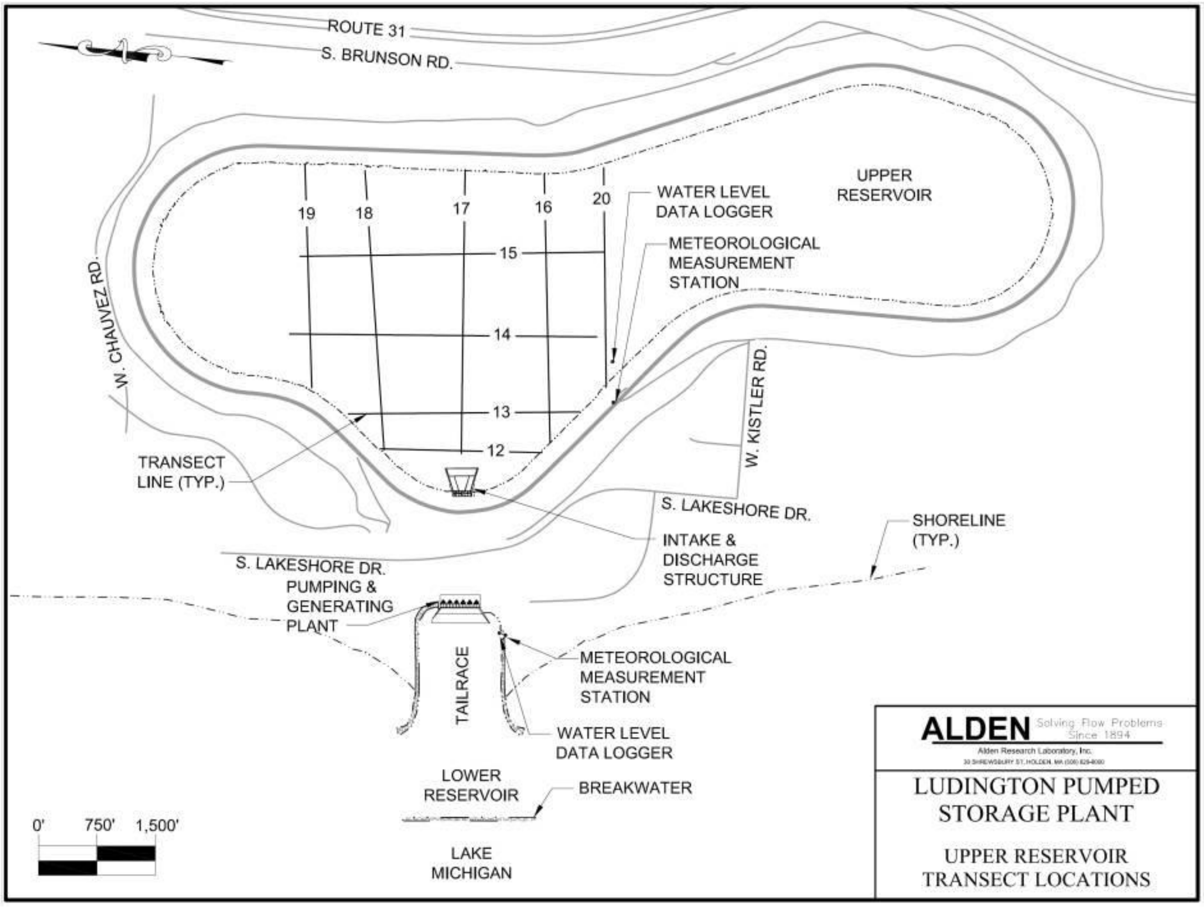

Figure 6 : Field Velocity Traverse Locations

4.1 Limitations of Field Velocity Measurements

Field velocity measurement data can be a very useful tool for calibration and verification of physical and CFD models. There are a few key points to understand when evaluating the field velocity data. First, the data is collected over a period of time, therefore velocity is not measured at the same time at each transect or even along each individual transect. Second, the bin resolution (bin size) can affect the data. The data is averaged over the entire bin, so small localized high-velocity regions could be averaged out of the data. The unsteadiness of the flow may not be completely captured since the data is for a short period of time.

5.0 2011 PHYSICAL MODEL STUDY

In 2011 a 1:93 physical scale model of the Ludington Pumped Storage Plant's upper reservoir was constructed at Alden to evaluate the hydraulic conditions at the reservoir that would result from the planned upgrades. A photo of the physical model is shown in Figure 7 while a close up photo of the intake structure is shown in Figure 8.

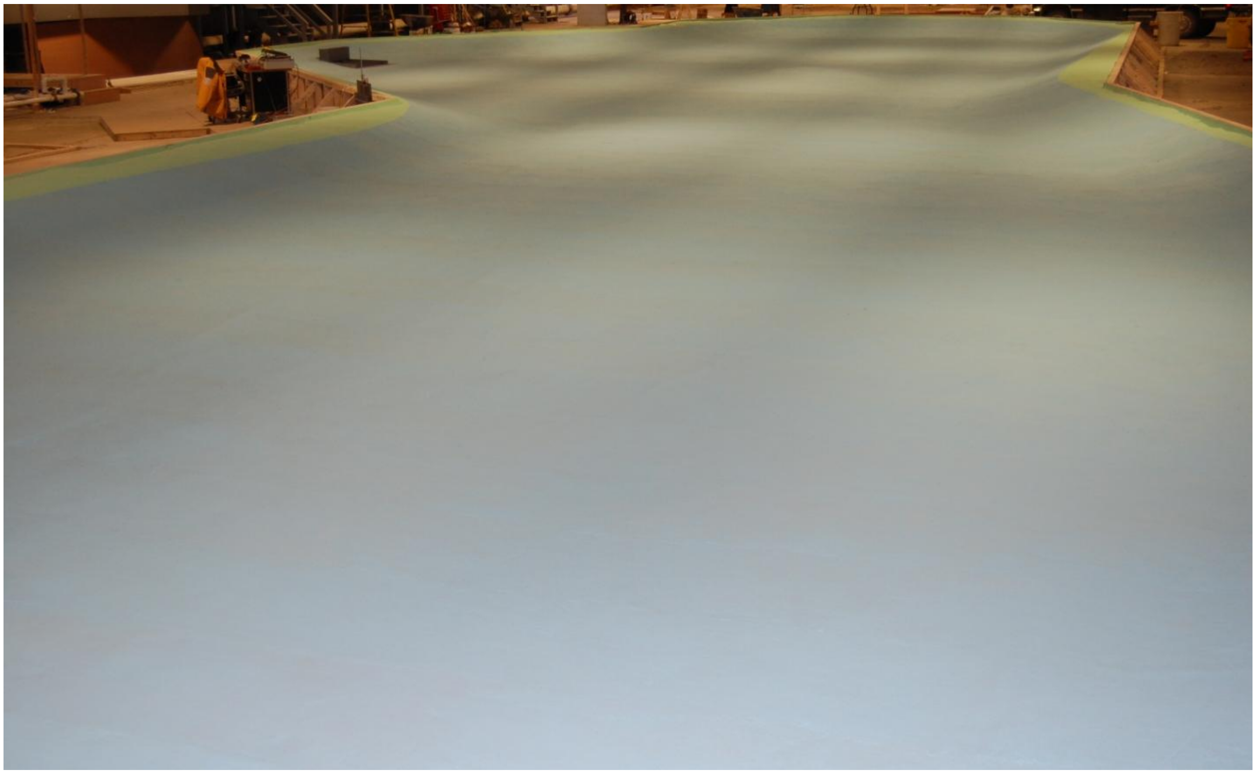

Figure 7 : Photo of the Overall Ludington Upper Reservoir Physical Model

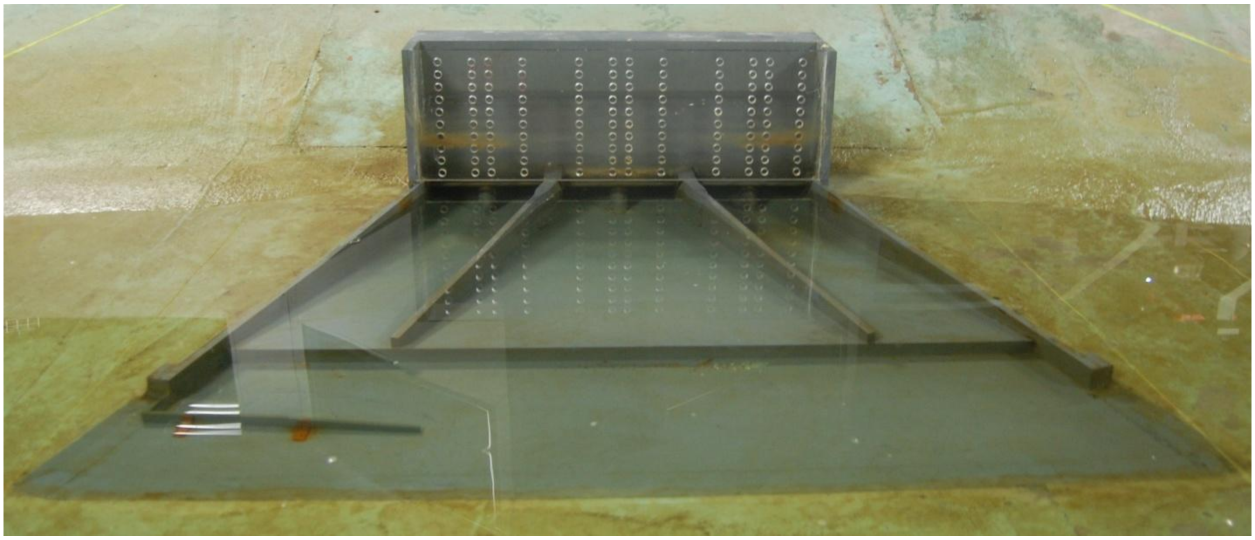

Figure 8 : Physical Model Intake\discharge Structure

5.1 Model Similitude

The 2011 model study used Froude similitude for correlating the model and prototype relationships. The model study was conducted utilizing the HIS modeling criteria and techniques to minimize the potential for scale effects at the intake.

Physical (hydraulic) model studies are governed by similitude requirements which must be followed to obtain meaningful results. As the flow to the intake\discharge structure is essentially a free surface flow, it is required that the dimensionless parameters that predominantly characterize flow patterns in free surface (open channel) flows should be the same in the physical model and the prototype. Other dimensionless parameters, such as the Reynolds number, must be high enough in the model to have negligible influence (scale effects) on flow patterns.

5.2 Free Surface Flow

Free surface flows (open channel flows) are governed by gravitational and inertial forces. The Froude number, represents the ratio of inertial to gravitational forces, and can be defined as:

$$F = \frac{V}{\sqrt{gL}} \tag{1}$$

where:

V = a characteristic velocity, such as average velocity at intake

g = acceleration due to gravity

L = a characteristic length such as hydraulic depth or hydraulic radius at the intake

For Froude number similarity, the Froude numbers of prototype and model are equal, therefore,

$$F_r = \frac{F_m}{F_p} = 1 \qquad (2)$$

where the subscripts m, p, and r denote model, prototype, and ratio between model and prototype, respectively.

The Reynolds number, Re, is the ratio of viscous to inertial forces and is defined as follows:

$$\mathrm{Re} = \frac{VL}{\nu} \qquad (3)$$

where, V and L are as defined previously, and:

ν = kinematic viscosity of the fluid.

By definition of the Reynolds and Froude numbers, if the model and prototype use the same fluid, it is not possible to equate both in the model and prototype. At high Reynolds numbers, viscous forces are negligible and the flow patterns are independent of the Reynolds number. For models involving the formation of vortices, it is important to select a geometric scale large enough to achieve a large enough Reynolds number and fully turbulent flow in the approach channel and the intake, however, it is not necessary to match the model and prototype Reynolds number. In addition, for free surface flows involving the possible formation of free surface vortices, additional similitude considerations are made as described below to minimize viscous and surface tension scale effects in the vicinity of the intake structure.

5.3 Vortex Formation

The 1:93 scale model showed vortices that were not observed in the 1969 model study. The physical model study showed a maximum of a Type 5 (HIS) free surface vortex entering the intake when the plant was generating even with the modifications derived from the previous study. Based on the 1969 study, these vortices were not observed in the final design. The difference in the results of these two studies is most likely due to the change in model scale and reduced scale effects in the 2011 model.

Field observations were subsequently made at the plant to verify the formation of vortices in the prototype. The location of the vortices matched well between the physical model study and the prototype, however the strength of the vortices in the prototype is not easy to determine because of the limited ability to view vortex strength in the field.

For operational reasons, there are occasionally short periods of time when some of the units are pumping while others are generating. These conditions were evaluated in the 2011 physical model study and it was determined that vortex formation increased when the plant was operated in this manner. The shear layer that developed between the pumping and generating flows created flow patterns conducive to vortex formation. Persistent Type 5 free surface vortices were observed in the 2011 model study for these operating conditions.

The 2011 model was subsequently utilized to develop a modified vortex suppressor design to reduce the strength of the vortices to Type 2 or less. The vortex suppressor that was developed as a result of this effort mitigated the surface vortex issue for all of the varying flow scenarios including the combined pumping and generating conditions. Due to the limited timeframes of the combined pumping/generating scenario, the vortex suppressor will only be implemented if operational needs change or performance issues are found following the upgrades.

5.4 Scour Potential

Scour potential in the physical model study was tested by measuring the near-bed velocities. The near-bed velocities from the 1969 physical model study were used for comparison to the current model study results. Velocities were also recorded at the upgraded flow rates to determine locations that may need additional scour protection from increased velocities. The locations investigated in the new model were selected based on the results of the CFD model. The areas of high shear stress in the CFD model were evaluated in the physical model. The location of the higher velocities in the CFD model matched reasonably well to physical model study results.

5.5 Limitations of the Physical Model Study

While the 2011 physical model study was an improvement on the methods used in 1969, there are still some limitations to the physical model. The data, in comparison to a CFD study, are more difficult to collect, especially in a dynamic flow field. The data in a physical model can be collected, but the time and cost of taking the measurements can become cost prohibitive compared to a CFD model. The physical model involves more labor and repeated testing to be able to measure all of the data that a CFD model can provide.

6.0 DIFFERENCES IN THE PHYSICAL MODEL RESULTS

Vortex formation in the 1969 model study showed that there were no air or debris entraining vortices, but the 2011 model study showed the presence of higher strength vortices. Extensive research since 1969 has resulted in a better understanding of the necessary model scales to capture vortex formation. This study found that the vortex suppressing baffle wall that was originally developed was not as effective at eliminating or reducing vortex strength as originally thought. This finding was confirmed by observations in the field. Similar studies comparing observed and predicted vortex formation at other structures lead to the present HIS guidelines on model scale. (Hecker and Larsen) The reason that the 1969 model under predicted the vortex formation appears to be the result of the Weber and Reynolds number for the model study being too low. Since the Weber and Reynolds numbers were too low there were likely scale effects that reduced the vortex formation. Essentially the surface tension effects in the 1969 model were too great for the vortices to overcome.

Applying the proper scaling criteria for a physical model allows for the accurate prediction of the flow patterns and hydraulics in the upper reservoir and approaching the intake structure. A properly designed physical model is capable of showing vortex formation at the intake, the flow patterns in the upper reservoir and the velocity magnitude of the flow.

7.0 CFD MODEL

The flow patterns and velocities in the upper reservoir during pumping and generating modes are transient and highly three dimensional, particularly in the areas near the intake structure. To properly simulate these flow patterns a fully three dimensional computational fluid dynamics (CFD) model was developed.

Selection of the CFD model depends on how the models handle the free surface and how the large variations in relevant length scales are accommodated. Variations in the free surface elevation within the model domain are large (about 67 ft) and the relevant length scales range from approximately 5 feet at the intake structure to hundreds of feet in the far reaches of the reservoir. In selecting an appropriate model, it is also important to note that all of the simulations must be run time dependent to properly model the changing hydrodynamics during pumping (filling) and generating (emptying).

FLOW-3D is a commercially available CFD software package particularly well suited for steady and unsteady simulations involving a severely deformed or time varying free surface. The model solves the fully three dimensional Navier-Stokes equations on a structured hexagonal grid. The location of the free surface in FLOW-3D is computed using the Volume of Fluid (VOF) method (Hirt and Nichols, 1980). This formulation consists of three parts: a scheme to describe the shape and location of the free surface, a method to track the evolution of the shape and location of the free surface through time and space, and a means for applying boundary conditions to the free surface. The simulations do not include the movement of the air above the water; it is assumed that the air has no significant effect on the water movement.

FLOW-3D uses the Fractional Area/Volume Obstacle Representation (FAVOR) method (Hirt and Sicilian, 1985) for the modeling of solid obstacles, such as topology and the intake structure. The FAVOR method allows complex shapes to be simulated without resorting to ‘stair stepping’ the boundaries. This method approaches the accuracy of more computationally intensive boundary fitted grids. The representation of the intake structure is shown in Figure 9. The computational mesh contained approximately 3 million cells.

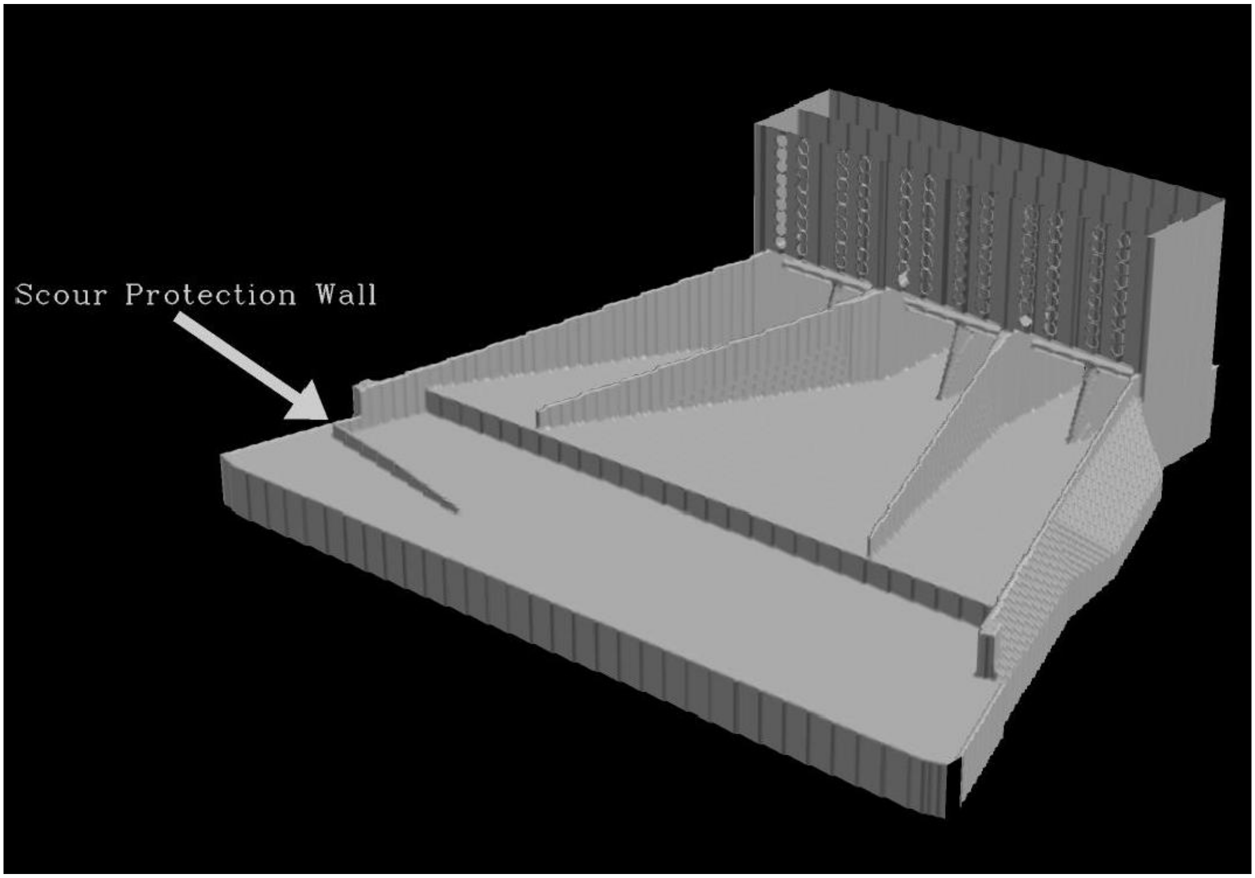

Figure 9 : Computational Model Representation of the Intake\Discharge Structure

7.1 Vortex Formation

CFD is presently not considered an appropriate tool to evaluate vortex activity. The computational requirements necessary to capture potential vortex activity are not cost effective. Additionally, CFD has not been shown to accurately represent vortex strength or persistence.

7.2 Scour Potential

All CFD simulations were performed to evaluate changes in gross reservoir flow patterns and surface velocity magnitudes. In addition, simulations modeling all units pumping at the upgraded flow rate were evaluated to predict velocity near the reservoir bottom. Increases in the velocity near the reservoir bottom could create an increased potential for scour of the reservoir liner. Areas of the reservoir where the model showed high velocity near the bed were useful to guide selection of the measurement locations in the physical model. The CFD model did not include the ability to simulate erosion of the liner and associated changes in the model geometry.

The simulations were run such that data could only be evaluated at three water surface elevations, reducing computational requirements. During intermediate water levels, as the water surface elevation rose during pumping or fell during generating, a coarse mesh was used to increase computational speed while maintaining gross flow patterns of the reservoir. Prior to reaching each water surface elevation of interest, the mesh resolution was increased and the simulation was run for 60 minutes prior to processing data.

For the purposes of this study, the method of determining potential areas of concern relating to scour was as follows:

1. Determined maximum CFD calculated shear stress outside of the existing riprap footprint at the existing flow capacity. This served as the maximum allowable shear stress for an unarmored bed.
2. Determined the maximum CFD calculated velocity magnitude outside the armored area at an elevation of 2 ft above the reservoir bed at the existing flow capacity. This served as the maximum allowable near-bed velocity magnitude, replacing the 2.5 ft/s used in the original riprap design of the upper reservoir. This value was used based on the observation that no scour has occurred in unarmored areas at the existing flow capacity.
3. For simulations of the upgraded capacity, areas outside of the existing riprap footprint with shear stress greater than the new maximum allowable shear stress were determined.
4. For simulations of the upgraded capacity, areas outside of the existing riprap footprint with near-bed velocity magnitudes greater than the new maximum allowable velocity magnitude were determined.

The maximum velocity magnitude outside of the existing riprap footprint and 2 ft above the reservoir bed was 4 ft/s. This velocity occurred at the mid pond elevation, as shown in Figure 10. This velocity was used as the limiting value for evaluation of flows at the upgraded flow rate. The areas of near-bed velocity greater than 4 ft/s at the upgraded flow rate are shown in

Figure 11; the existing riprap footprint is also shown in this same figure. The maximum near-bed velocity outside of the existing riprap footprint was 6 ft/s.

The CFD model found that the maximum velocity outside of the existing riprap footprint, for existing plant flows, was above the design specification used to develop the liner protection.

The near-bed velocity and scour calculated by the CFD model were used to guide the physical model testing. The available data from the CFD model made the simulation results a good tool for screening potential areas of increased scour. In addition to the CFD model, the physical model was used to evaluate scour because of uncertainty associated with the calculated values from the simulation results. The calculated near-wall velocity is dependent on the wall function model employed by the software to produce the velocity gradient at the boundary. Additionally, as discussed below, the model used hydraulically smooth boundaries.

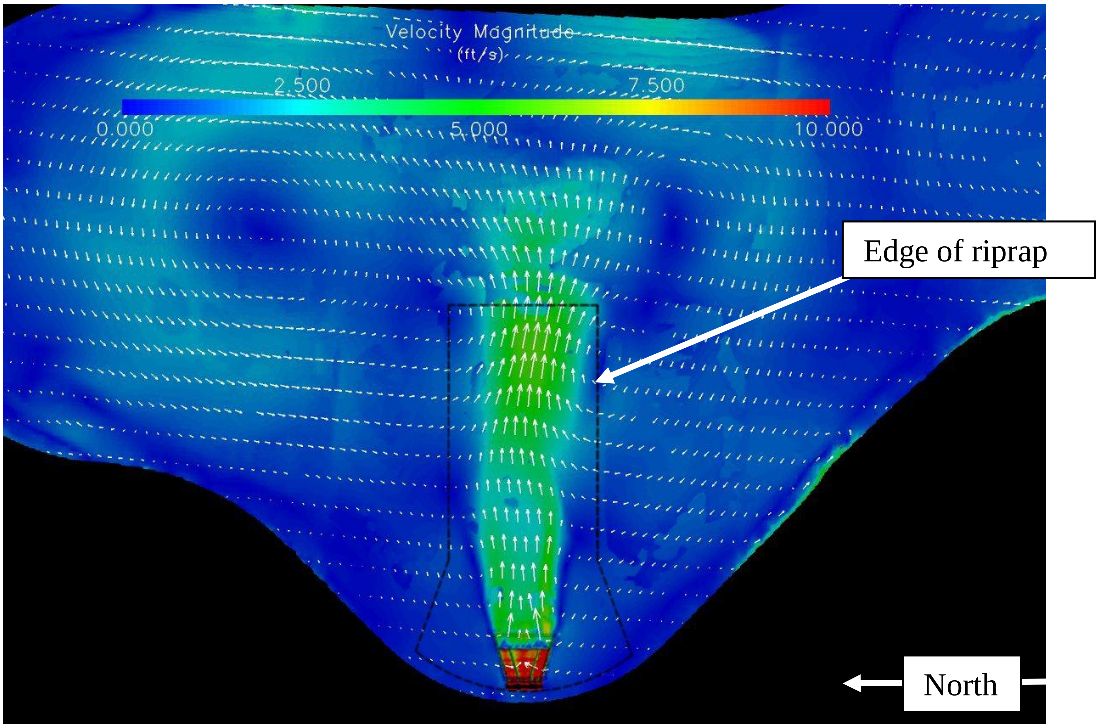

Figure 10 : Near-Bottom Velocity Magnitude, Existing Flow Rates, Mid Pond

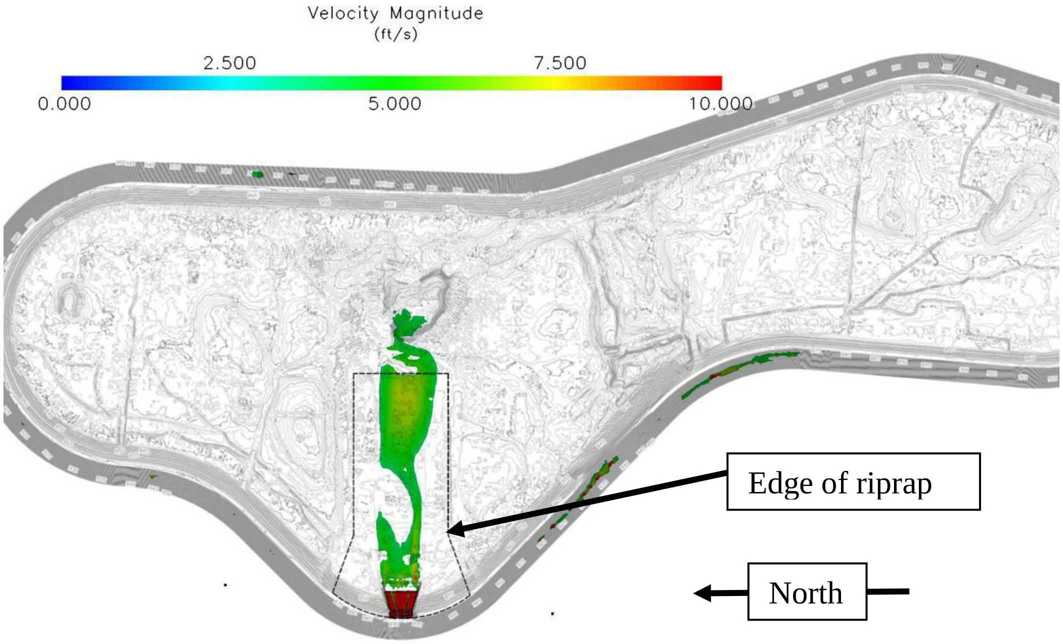

Figure 11 : Near-Bottom Velocity Magnitude Exceeding Maximum Velocity Magnitude, Upgraded Flow Rates, Mid Pond

The locations identified in the CFD model were subsequently evaluated further utilizing the 1:93 scale physical model study. The velocity data from the physical model and the CFD model were comparable and only small differences in the velocity were measured in the physical model.

7.3 Limitations of CFD Models

All modeling has some limits effecting what can be ascertained from the results. Some of the limitations of CFD modeling for this project are discussed below.

Wind shear was not replicated in the model. Wind shear was not considered a factor driving flow patterns during normal operations. Since the data from the CFD model was compared only against other CFD runs, this was considered to be a reasonable assumption.

All of the model boundaries and structures, including the reservoir bed and the concrete intake structure, are hydraulically smooth. The absolute roughness of the concrete and reservoir bed material (including riprap) is very small relative to the overall size of the structures and the model domain, making the assumption reasonable. Furthermore, for this application, this assumption is conservative since the hydraulically smooth boundaries will tend to result in a slightly higher water velocity.

Simulation results were not used to determine absolute values of near-bed velocity and bed shear. Instead, the CFD results were used to guide the physical model data collection program. Data was not available to calibrate or validate the CFD model for the geometry (future condition) and grid resolution used; additionally the model used hydraulically smooth boundaries, which slightly increases the near-wall velocity. Without validation data, the uncertainty associated with the near-bed velocity made the simulation results unsuitable for updating the scour protection. In the absence of a physical model, the CFD results can be used to develop scour protection by accounting for model uncertainty and using a larger safety factor in the design.

Although numeric model results were used to determine general circulation patterns near the intake structure, the results were not used to evaluate vortex activity. CFD is not an appropriate tool for evaluating the potential for vortex formation, nor is it, recognized by the Hydraulic Institute of Standards as an appropriate tool for such.

8.0 COMPARISON OF CFD MODELS TO PHYSICAL MODELS.

There are a few main differences between a CFD model and a physical model study. The differences are based on how the CFD model solves the flow field and how data can be collected from a physical model.

The CFD model allows for data measurements to be recorded at all locations simultaneously. Measuring data at a spatial resolution similar to the CFD model grid resolution would take a great amount of time and making the measurements simultaneously would be impractical in a physical model. While the CFD model can produce a significant amount of data, because of the continuously changing water surface the resolution of the model used for Ludington was manipulated to improve the processing time required. As a result, the CFD model was only able to produce high resolution results for specific elevations in the reservoir. More elevations would have been possible, however the cost and time associated with the additional computational time would not have appreciably increased the knowledge produced by the study. The CFD model was excellent at providing streamlines of the flow in the reservoir. The streamlines made for a better visual understanding of the hydraulics in the reservoir.

The physical model captures the unsteady nature of the flow patterns in the reservoir. The model accurately replicates the jets that emanate from the intake structure. Jets will inherently wobble and not discharge into a consistent area. Although CFD captures the jet discharge, the instability is introduced based on turbulence closure models. Selection of the turbulence closure model and model parameters can affect the accuracy of the jet instabilities.

9.0 CONCLUSION

Hydraulic modeling to achieve a greater understanding of the flow patterns around a hydraulic structure can be an invaluable tool. A comparison of the field velocity data to the CFD and physical model velocity data shows that both models accurately predict the flow fields. The physical model will capture the unsteady phenomenon while the CFD model makes it easier to visualize the flow patterns and understand the hydraulics. The combination of the CFD model and the physical model allowed powerful approaches to evaluate flow parameters of concern. The combination of both modeling techniques optimizes usage of the modeling budget and provides results with the highest degree of confidence.

Whether the model is computational or physical, having an understanding of the benefits and limitations of both types of modeling will allow for a modeling study that provides the most pertinent information. These studies provided information that prevented the reservoir from scouring and provided a method for dissipating the observed vortices. The design information provided by the 1969 model study allowed the reservoir to essentially operate for approximately 40 years without any maintenance issues attributable to vortex activity or scour.

The current model study was conducted to evaluate the impact of the upgrade of the Ludington units on the upper reservoir and intake structure. The CFD and physical models played a complementary role to each other. The CFD model provided data that helped to locate areas of concern and allowed for fewer measurements to be required in the physical model. The physical model provided insight into phenomena that the CFD model couldn’t, such as vortex formation at the intake structure and the unsteadiness of the jet created from pumping (filling) operations.

The testing that was conducted for the Overhaul project provided confidence in the implementation of the unit upgrade. The results of the study showed those areas with the greatest potential for increased scour were and the geometry of a vortex suppressor (if needed). Based on the results of the study and the engineering analysis of the results, the Ludington Pumped Storage Plant will continue to operate without any major problems.

References:

Jain, A.K., Raju, K.G.R., and Garde, R.J., "Vortex Formation at Vertical Pump Intakes," ASCE Journal of Hydraulics Division, Vol. 104, 1978.

Daggett, L. and Keulegan, G.H., "Similitude in Free-Surface Vortex Formations," Journal of the Hydraulics Division, ASCE, November, 1974.

Hecker, G.E. and Larsen, J., "Model to Prototype Comparison of Vortices for a Pumped Storage Intake," Proceedings, Joint ASME/ASCE Mechanics Conference, Boulder, CO, June, 1981.

Hydraulic Institute Standards, American National Standard for Pump Intake Design, ANSI/HI 9.8-1998

Nichols, BD., Hirt, C.W. and Hotchkiss, R.S. (1980) “SOLA-VOF: A Solution Algorithm for Transient Fluid Flow with Multiple Free Boundaries”, Los Alamos Sci. Lab. Report LA-8356.

Hirt, C.W., and Sicilian, J.M. (1985). “A porosity technique for the definition of obstacles in rectangular cell meshes.” Proc., 4th Int. Conf. Ship Hydro., National Academy of Science, Washington, D.C., 119.

Timothy C. Sassaman is an engineer at Alden Research Laboratory, Inc, a leading physical and computational flow modeling laboratories in North America. He is responsible for physical hydraulic models to evaluate flow related issues. Experience includes modeling civil hydraulic structures and nuclear safety issues.

Mr. Andrew Johansson is the Director of Hydraulic Modeling for Alden with over 20 years of experience in hydraulic modeling. He obtained his B.S. degree in Mechanical Engineering in 1995, from Worcester Polytechnic Institute. Mr. Johansson provided day to day oversight of lab operations and manages all aspects of model studies conducted for electric power utilities, architect-engineering firms, equipment manufacturers and governmental agencies.

Mr. Ryan Jones is an engineer for Alden. He obtained his B.S. degree in Mechanical Engineering in 2006, from the University of Denver, and his M.S. degree in Civil and Environmental Engineering in 2010, from the University of California, Davis. Mr. Jones' expertise lies in Computational Fluid Dynamics (CFD) modeling of complex free surface and closed-conduit flows.

Marianne Walter is a civil engineer in Consumers Energy's Engineering Services Department. Mrs. Walter joined Consumers Energy in 2009 and has worked on numerous projects in both fossil and hydro power plants. She served as the Project Engineer for the Ludington Pumped Storage Plant Numeric and Hydraulic Modeling Project. In this role, she had primary responsibility for administrative and technical oversight of the test program and results.