

Utah State University

DigitalCommons@USU

International Junior Researcher and Engineer
Workshop on Hydraulic Structures

Jun 17th, 12:00 AM - Jun 20th, 12:00 AM

Hazards at Low-Head Dams

Riley J. Olsen

Michael C. Johnson

Follow this and additional works at: <https://digitalcommons.usu.edu/ewhs>

 Part of the [Civil and Environmental Engineering Commons](#)

Olsen, Riley J. and Johnson, Michael C., "Hazards at Low-Head Dams" (2012). *International Junior Researcher and Engineer Workshop on Hydraulic Structures*. 4.
<https://digitalcommons.usu.edu/ewhs/Sessions/2/4>

This Event is brought to you for free and open access by the Conferences and Events at DigitalCommons@USU. It has been accepted for inclusion in International Junior Researcher and Engineer Workshop on Hydraulic Structures by an authorized administrator of DigitalCommons@USU. For more information, please contact digitalcommons@usu.edu.



HAZARDS AT LOW-HEAD DAMS

Riley J. OLSEN

Civil and Env. Engineering, Utah State University, USA, riley.j.olsen@gmail.com

Michael C. JOHNSON

Civil and Env. Engineering, Utah State University, USA, michael.johnson@usu.edu

ABSTRACT: Low-head dams are small structures that are built for many purposes, the most common being to impound small amounts of water for various uses. At certain flow conditions, a dangerous countercurrent, known as a roller, can form downstream of a low-head dam. This current possesses an upstream directed surface velocity that can effectively trap debris, as well as unsuspecting humans, at the downstream side of the dam. It is this current that is responsible for the deaths of many individuals that have ventured too close to these structures over the years. It is the objective of this study to identify a relationship between easily observable parameters and the roller strength that can be used as a classification system. This will be done primarily through the use of computational fluid dynamics software to simulate various flow conditions. The results of these numerical models will then be compared, and a relationship will be identified. This study is currently underway, and therefore conclusive results are not available at this time. Although, comparison of the numerical results to physical model results have been used to verify that the flow conditions being produced by the software accurately represent the physical flow conditions.

Keywords: low-head dam, drowning machine.

INTRODUCTION

Low-head dams have been constructed historically to serve a wide variety of purposes. Some are meant to impound small volumes of water to be used for irrigation and cooling of power plants, supply water to municipalities and industry, and simply to provide for recreational activities. Others are in place to house and protect utility lines at river crossings. Some are built to enhance water quality downstream through the entrainment of air into the water. They can take many different forms, with the most common being the flat-topped and the ogee crested weir.

As water flows over a low-head dam or drop structure in a mildly sloped channel, the

flow regime smoothly transitions from subcritical to supercritical. As it continues past the dam, the flow must eventually return to subcritical flow at a distance downstream depending on the slope of the channel and the tailwater conditions. A transition from supercritical to subcritical flow is not a smooth one as it is when going in the opposite direction. Instead, a hydraulic jump is formed where the transition takes place, with the purpose of dissipating the excess energy possessed by the high velocity supercritical flow.

When flow conditions at a low-head dam cause the hydraulic jump to be submerged, a strong current can be formed at the downstream face of the dam that features a characteristic upstream directed surface velocity. This current, which is commonly referred to as a “hydraulic” by recreational water users or “roller” by the engineering community, can often catch debris and hold it near the face of the dam for long periods of time. Occasionally, an unsuspecting person can get too close to one of these structures and find themselves incapable of escaping the relentless current, often struggling to the point of exhaustion and ultimately drowning before being rescued.

Several studies have been performed on the subject of low-head dams, primarily focusing on the dangerous hydraulic conditions and possible remediation techniques that have been proposed. Some of the more significant studies include those performed by LEUTHEUSSER (1988), LEUTHEUSSER and BIRK (1991), HOTCHKISS and COMSTOCK (1992), and LEUTHEUSSER and FAN (2001). These studies have done an excellent job of identifying the dangerous flow patterns created at low-head dams and suggesting possible solutions to the problem, although it seems little has been done when it comes to classifying the hazards present based on easily obtainable parameters.

In their study, LEUTHEUSSER and FAN (2001) utilize a submergence factor (S) of the hydraulic jump to identify key transition points in the formation of the roller. This submergence factor, as presented in Eq (1), requires the depth of the subcritical sequent depth of the optimum hydraulic jump (Y_2), which can be difficult to determine. Y_4 in Eq. (1) is the downstream tailwater depth.

$$S = \frac{Y_4 - Y_2}{Y_2} \quad (1)$$

This study aims to identify a relationship between easily observable and measurable parameters that will allow low-head dams to be classified according to the danger encountered

by the public. This classification system would help recreational water users and dam owners alike assess hazards and act appropriately in terms of safety and liability.

RESEARCH OBJECTIVES AND METHODOLOGY

This study, which is currently underway at the Utah Water Research Laboratory (UWRL) in Logan, Utah, will use the computational fluid dynamics (CFD) software Flow-3D™ to numerically model flow over low-head dams at various flow conditions. Two dam shapes are being examined as a part of this research: the flat topped weir and the ogee crested weir. Dam heights (P) being tested are 0.61 m, 1.52 m, and 3.05 m, with varying upstream and downstream water depths.

The setup of the numerical models in Flow-3D™ will be identical besides the upstream water depth (h_u) and downstream water depth (h_d). Simulations are being run in series, with a single series consisting of a particular dam size and h_u held constant, while h_d is increased incrementally in each simulation. Once a series of simulations has been completed, a new series is set up using a different value of h_u .

The boundary conditions of the computational domain used will model actual flow conditions at a low-head dam as closely as possible, and will be kept the same for each simulation. The mesh boundaries will be set as follows:

- Upstream boundary (X_{min}): Specified pressure boundary with a stagnation pressure set to zero and specified fluid height
- Downstream boundary (X_{max}): Specified pressure boundary with a stagnation pressure set to zero and specified fluid height
- Bottom boundary (Z_{min}): Wall boundary (no slip)
- Top boundary (Z_{max}): Symmetry boundary (no influence on model due to open channel)
- Side boundaries (Y_{min} and Y_{max}): Wall boundaries (no slip)

Once a numerical simulation has been completed, the flow rate, minimum surface velocity in the direction of flow (negative being upstream directed), water surface elevations at a distance of $2P$ upstream and $3P$ downstream from the upstream face of the dam, and Froude Numbers at these same locations are extracted from the results. A distance of $2P$ is used for the upstream

water depth measurement location to avoid the effects of drawdown as water flows over the dam. A distance of $3P$ will not be used as the downstream measuring location to avoid turbulence from the hydraulic jump and the associated error. These distances will be used as the standard for this classification system. A three-dimensional animation of the simulations will also be created from the CFD results. This animation will show each cell of the computational domain represented by color based on the magnitude of the x-velocities. A definition sketch of the numerical model setup and physical model setup of a flat topped weir simulation is shown in Figure 1.

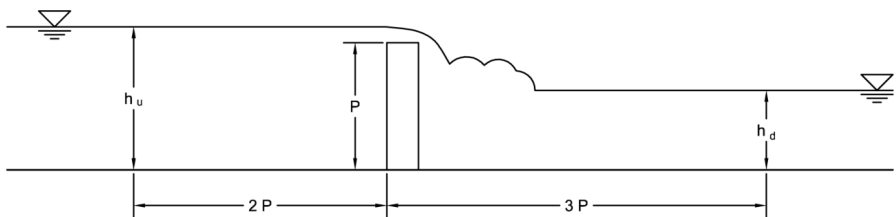


Figure1 – Numerical model setup

In order to verify the accuracy of the numerical results obtained through the CFD program, physical models of several of the simulations will be built and tested at the UWRL, utilizing a gravity fed rectangular laboratory flume (1.83 m x 9.14 m x 1.22 m deep). The physical models will be set up so that the water depths at the specified distances upstream and downstream of the dam match those at the same distances in the corresponding CFD models as closely as possible. Once these water surface elevations are achieved, a flow meter will be used to measure the flow rate. This flow rate, as well as photos and video of the physical flow, will be compared to the numerically obtained flow rates and the CFD animations to verify that the physical process is being accurately reproduced by the CFD setup.

In addition to the verification of the CFD program, the physical model will also be used to test the roller's ability to trap a scaled human model. The human model was cut out of a sheet of high-density polyethylene (HDPE) and modelled to weigh exactly 85.3 kg at a one-fifth scale (0.68 kg model scale). In order to achieve this weight, material was cut out of the chest of the human shaped model. In order for the model to float in the upright position such as a human wearing a life jacket, bolts were attached to the ankles of the model and polystyrene foam filled the hollowed chest cavity. These modifications yielded a test subject that floated upright in the

water with everything below the shoulders being submerged.

RESULTS AND DISCUSSION

Because this study is still underway, data is still being collected and results are not being disclosed at this time. Although there is still much work to do in this study, several pairs of numerical and physical models used to verify the CFD setup accuracy have been completed. The results of these tests are shown in Table 1. Included are the flow rate, Q , upstream water depth, h_u , and downstream water depth, h_d , of both the numerical and physical models tested. Also included are the corresponding percent differences between these parameters.

Table 1 – Comparison of CFD results to physical model data

		Q (m ³ /s)	h_u (m)	h_d (m)
Comparison 1	CFD	0.064	0.686	0.305
	Physical Model	0.062	0.686	0.290
	% Difference	3.7%	0	5.3%
Comparison 2	CFD	0.064	0.686	0.387
	Physical Model	0.062	0.686	0.381
	% Difference	3.7%	0	1.6%
Comparison 3	CFD	0.303	0.805	0.677
	Physical Model	0.295	0.808	0.649
	% Difference	2.9%	0.4%	4.2%

As can be seen from the data in Table 1, the largest percentage difference in flow rates between a pair of simulations was 3.7% in both comparison 1 and 2. The average percent differences are as follows: 3.4% in flow rates, 0.13% in h_u , and 3.7% in h_d . These small discrepancies have been deemed acceptable for this study, and it has therefore been concluded that the CFD program and setup being used is accurately replicating physical flow conditions at low-head dams. Figures 2 and 3 show a snapshot of the CFD simulation and a photo of the physical model of comparison 2, respectively.

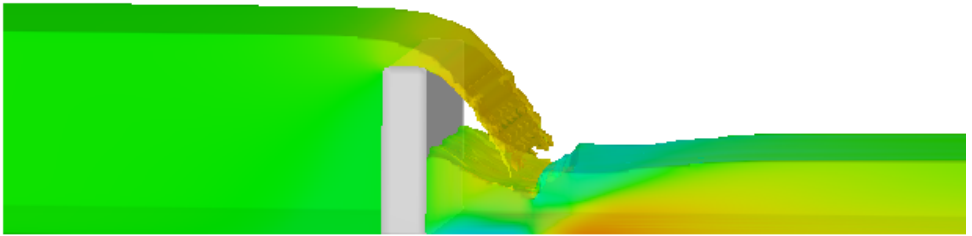


Figure 2 – Animation of CFD model from comparison 2



Figure 3 – Photo of physical model from comparison 2

CONCLUSION

The objective of this research is to increase the understanding of the dangerous hydraulic conditions that can be present at low-head dams at certain flow conditions. It is also hoped that a clear relationship between parameters easily measured and obtained in the field and roller strength can be identified and used as a classification system. This type of system would help recreational water users and dam owners assess dangers and therefore make better decisions in regard to safety and liability.

Because the study is currently in progress and there is still much data to collect and analyze, no conclusions have been made as yet. The data that has been collected and analyzed up to this point shows promising potential for achieving the desired outcomes.

LIST OF SYMBOLS

h_d = Downstream water depth

P = Dam height

Q = Flow rate

S = Submergence factor

Y_2 = Sequent subcritical depth of optimum hydraulic jump

Y_4 = Tailwater depth

REFERENCES

- HOTCHKISS, R. H., and COMSTOCK, M. (1992). "Discussion of 'Drownproofing of low overflow structures,' by H. J. Leutheusser and W. M. Birk." *J. Hydr. Eng.*, ASCE, 118(11), 1586-1589.
- LEUTHEUSSER, H. J. (1988). "Dam safety, yes. But what about safety at dams?" Proc., 1988 Nat. Conf., Hydr. Div., ASCE, Colorado Springs, CO, 1091-1096.
- LEUTHEUSSER, H. J., and BIRK, W. M. (1991). "Drownproofing of low overflow structures." *J. Hydr. Eng.*, ASCE, 117(2), 205-213.
- LEUTHEUSSER, H. J., and FAN, J. J. (2001). "Backward Flow Velocities of Submerged Hydraulic Jumps." *J. Hydr. Eng.*, ASCE, 127(6), 514-517.