Modeling Dams with Computational Fluid Dynamics: Past Success and New Directions

Jeff Burnham, PE, LEED AP

ABSTRACT

Computational Fluid Dynamics (CFD) is an emerging field that can be applied to a wide class of hydraulics and dam safety problems. Much work has been done in the past with modeling the hydraulics of dams and rivers, and recent hardware and software developments allow exploration of more complex phenomena such as sediment scour. This paper is divided into four sections. The first section reviews case studies of the five most common CFD dam and river simulations using the commercial software package FLOW-3D. The five most common simulations are spillway flow analysis, nappe impingement analysis, pressure/shear distribution and tractive force analysis, tailrace and stilling basin analysis, and river reach and intake analysis. The review is thorough and summarizes 96 case studies. The second section interprets the case studies and draws generalized conclusions regarding efficient dam and river modeling with FLOW-3D. The third section discusses cutting-edge topics in CFD modeling of dams and rivers: studying transient phenomena associated with gate motion, approximating sediment erosion and deposition, modeling cavitation formation and damage, and simulating air entrainment. The discussion includes summaries of 17 published case studies, difficulties encountered, software and hardware limitations, and the relation of CFD modeling to physical modeling. The final section presents original research comparing numerical models to experimental data for flow and scour in compound channels with abutments. The results are discussed as they relate to the generalized conclusions developed from the literature reviews.

INTRODUCTION

Computational Fluid Dynamics (CFD) covers a wide range of methods by which computers numerically simulate fluid flow. In CFD projects, the environment to be studied is virtualized and simplified into a digital model. The model includes numeric representations of the material elements (existing and proposed structures and topography) as well as methods of solving physical equations that simulate the motion of the flow. These models, and the results they produce, are visually displayed on the computer and allow the model to be reviewed without any physical analogue.

CFD models can be one, two, or three-dimensional. Digital modeling of open-channel hydraulics began in the 1980s with open-source, one-dimensional solutions of steady-state surface profiles published by the U.S. Army Corp of Engineers (USACE), and matured to include a variety of 1-D, 2-D, and 3-D open-source and for-profit software solutions by the 1990s. Perhaps the most famous one-dimensional package is HEC-RAS, published by the US Army Corp of Engineers Hydraulics Engineering Center. Two- and three-dimensional programs include commercial products by ANSYS, DHI, Flow Science, and CD-adapco, as well as freeware and open source products such as those published by OpenCFD Ltd.

A brief description of the FLOW-3D software package is included below, followed by a lengthy section examining the history of use of FLOW-3D for hydraulic modeling of dams and rivers. Each major research article and case study is described with a brief synopsis of results. The purpose of this lengthy review is to aid the interested user of the software who may lack the time to search the literature thoroughly. A second section discusses general approaches illustrated by the case studies, and a third section describes more recent developments in CFD-aided dam design and analysis, including modeling transient effects from gate motion, modeling sediment scour and deposition, modeling cavitation phenomena, and modeling air entrainment processes. A fourth section presents a comparison between CFD modeling and physical modeling of hydraulics and erosion in compound channels.

A BRIEF DESCRIPTION OF FLOW-3D

FLOW-3D solves the Navier-Stokes system of equations in three dimensions to simulate the flow of fluid. Two-fluid and one-fluid solvers are available, both of which use a proprietary volume-of-fluid (VOF) method to track the free surface. Trademarked as TruVOF, this approach is one of the defining features of the software and provides three important functions for free surface flow: accurate location and orientation of free surfaces within computational cells, accurate tracking of free surface motion through cells, and an accurate boundary condition applied at the free surface interface. The VOF technique was developed mathematically by the founder of Flow Science and was first described by Nichols and Hirt (1975) and more thoroughly detailed by Hirt and Nichols (1971). The importance of TruVOF when simulating free-surface flows for dam and river simulations are described by Cook and Richmond (2001), Teklemariam et al. (2002), and Rodriguez et al. (2004).

The software includes many optional models that add to or modify the basic Navier-Stokes equations. Additional models that are used frequently in hydraulics include options for describing the effects of turbulence, surface tension, heat transfer, fluid solidification, sediment scour, Lagrangian particles, granular flows, moving solids, solid deformation, air entrainment, cavitation, and porous media.

While TruVOF and its ability to track free surfaces is one of FLOW-3D's strongest attributes, one of the software's known weaknesses is in modeling boundary layer dynamics because of the assumptions that are necessary when applying an orthogonal mesh to non-orthogonal geometry. The simplification introduced is usually not an issue for large-scale problems such as those represented by rivers and spillways, because the assumptions (fully-developed turbulence, for example) hold true. For flow around very thin geometries, such as turbine blades or aircraft wings, a better CFD solution is often one that uses a body-fitted, non-orthogonal mesh. As evidenced by the literature review below, most industrial-scale hydraulics applications are well-suited to the benefits of the free-surface representation, and are not adversely affected by simplifications in the boundary layer dynamics.

LITERATURE REVIEW: APPLICATIONS OF FLOW-3D TO DAM & RIVER PROBLEMS

An English-language literature review of applications of FLOW-3D to dam, bridge, and river design and analysis is given below. The citations and descriptions are divided into sections by the general topic of the study. Multi-focus studies may be included in more than one section. The CFD software used is FLOW-3D, unless otherwise indicated. The studies are organized by topic:

- Spillway flow analysis:
    - submerged and unsubmerged ogee spillways,
    - standard and non-standard Parshall flumes,
    - partially and fully-submerged gated openings,
    - labyrinth spillways,
    - stepped spillways,
    - tunnels, pipes, and draft tubes.
- Nappe impingement analysis
- Pressure/shear distribution and tractive force analysis
- Tailrace and stilling basin analysis
- River reach and intake analysis

An additional literature review of sedimentation and rock transport applications of FLOW-3D is given later in the review of “cutting edge” applications of FLOW-3D.

Spillway Flow Analysis

- Simulations of free discharge and submerged ogee spillways:
  - A series of 2-D unsubmerged ogee spillway models demonstrated a relative flow rate error of less than ±1% compared to physical model data when non-dimensional H_e/H_d (effective head divided by design head) is greater than 0.7, and less than ±6% when H_e/H_d is greater than 0.2 (Savage and Johnson 2001).
  - A series of 3-D ogee spillway model with piers showed a relative error of ±1.7% when dimensionless static head H/H_d = 1.33, ±0.1% when H/H_d = 1.00, and ±3.9% when H/H_d = 0.50 (Ho et al. 2003b).
  - A series of 2-D ogee spillway models comparing numerical results to a simplified empirical equation found relative over-prediction of discharges by 17% when dimensionless static head H/H_d = 1.33, 16% when H/H_d = 1.00, and 10% when H/H_d = 0.50. The study authors concluded that the over-prediction was due to modeling the flow as inviscid, thus underestimating the fluid drag (Ho et al. 2001, 2003a).
  - A 3-D ogee spillway model of the main spillway at Buffalo Dam, Victoria, including half of the central bay, one full outer bay, and one Type-2 round-nose pier showed a difference between US Army Corp of Engineers (USACE) empirical equations of ±3.0% when H/H_d = 1.00 and ±0.5% when H/H_d = 1.33 (Ho et. al 2003b).
  - A series of 2-D submerged ogee spillway models reported over two publications demonstrated a maximum relative flow rate error of 3.2% compared to physical model data for both submerged and unsubmerged flip-bucket and horizontal-apron spillways. The average flow rate disagreement magnitude of all thirteen runs was less than ±1.5% (Johnson and Savage 2006, Savage and Johnson 2009).
  - A series of simulations compared 3-D numerical model output to physical model data for free discharges over ogee spillways at the Wuskwatim dam on the Burntwood River, Manitoba (spillway-height-to-design-head ratio P/H_d = 0.9, with 7 headwater levels HWLs simulated), Limestone dam (P/H_d = 1.4, 8 HWLs), and Conawapa dam (P/H_d = 1.8, 9 HWLs). Water surface profiles were found to have good agreement, and simulated discharges were within the experimental error of the physical models for higher headwater levels (average error for all runs was less than ±6%). The CFD models tended to overestimate discharge relative to the physical model for P/H_d ratios less than approximately 1.6, and underestimate discharge when P/H_d increased (Chanel and Doering 2007, 2008).
  - A “near field” numerical model of existing and proposed Floodway Outlet Structures at Red River Floodway, Winnipeg, Manitoba showed excellent agreement with photographic and anecdotal observations of the existing spillway, as well as with results of scaled physical modeling. The numeric model correctly predicted the observed water levels and presence and magnitude of waves. The model was used to analyze proposed dimensions, configurations, and designs of the outlet structures to pass a 130% increase in design flow (Gendzelevich et al. 2009).
  - 3-D numerical models were used to develop new discharge capacity rating curves for increased design heads at the Boundary Dam spillway on the Pend Orielle River in Washington. The crest design capacity was increased by 100%, and the 3-D analyses provided the designers with a high degree of confidence in the proposed design (McPhail et al. 2010).

- Simulations of standard and non-standard Parshall flumes:
    - A series of numerical models of free-discharge and submerged standard Parshall flumes compared CFD model results to empiric and analytic equations in the US Bureau of Reclamation Water Measurement Manual (1984) and revised equations by Peck (1988). Using a Large Eddy Simulation (LES) turbulence model, the results for free-discharge and moderately submerged flumes were within 3% of the empirical relations (Hirt and Williams 1994).
    - A series of numerical models were developed to determine correction procedures for 2-ft to 8-ft Parshall flumes with non-standard wingwalls and when staff gauge or centerline measurements are not taken at the standard location. Taken together, these conditions previously could result in up to 65% measurement errors, and the correction procedure reduces the measurement error to 5% (Heiner and Barfuss, 2011).
- Simulations of partially and fully-submerged gated openings:
    - A 2-D numerical model was developed to examine free-surface and fully submerged hydraulic jumps downstream of a partial sluice gate opening. It was determined that the jump location can be controlled, and flow losses minimized, by an abrupt drop downstream (Richardson 1996).
    - Two test cases of a 3-D numerical model were developed to validate flow through submerged diversion ports at the Conawapa Generating Station on the Nelson River, Manitoba. Simulation rating curves accurately matched the physical model results (Teklemariam et al. 2002).
    - A 3-D numeric model of submerged, gated sluiceways with piers and downstream stoplogs and guides was developed for the Wuskwatim Dam project on the Burntwood River, Manitoba. The hydraulic performance of the proposed spillway was validated against physical model studies, and simulation results allowed the designers to raise the spillway crest and reduce stoplog requirements (Teklemariam et al. 2002).
    - A recalibration of the rating curve of the spillway at Hume Dam, NSW, was performed using a 3-D simulation to determine the effect of an overtopped bridge above the spillway. The numeric model showed good agreement against a 1:50 scale model for the studied case of orifice flow between the spillway bay and bridge (Ho et. al 2003b).
    - Six additional studies were undertaken by Ho et. al (2006) for gated spillways: two for the Warragamba dam, New South Wales (drum gate and radial gate bays), one each for the Buffalo, Wivenhoe, and proposed Tallowa dams (Victoria, Queensland, and New South Wales, respectively), and one for the Goulburn weir (Victoria). 3-D numerical model results were compared to results of physical models ranging in scale from 1:50 to 1:100. Six of the models were developed to reevaluate discharge curves and three models included transitional and orifice flow. The discharge rate of the numeric models tended to over-predict the physical model data by up to 5%, with average over-prediction of about 3%, for heads equal to and greater than the spillway design head.
    - A series of simulations compared 3-D numerical model output to physical model data for partially-open gated discharges for the Wuskwatim spillway, Limestone spillway, and Conawapa spillway. 2-, 4-, and 6-m openings were studied for the Wuskwatim and Limestone cases, while 1-, 3-, and 5-m openings were analyzed for the Conawapa case. Several HWLs were modeled for each opening. Nested mesh blocks were used to successively reduce cell sizes by factors of two until the change in discharge was less than 1%. For the Wuskwatim case, the CFD model very closely reproduced the 6-m opening discharge, slightly overestimated discharge (by less than 2%) for the 4-m opening, and significantly overestimated discharge for the 2-m opening. This last result suggested to the

authors the need for a smaller cell size in the region of the opening when the cell size was large. The Limestone spillway case produced similarly close results. The 3- and 5-m opening Conawapa results were mostly within the range of experimental error. The 1-m opening Conawapa case was the least successful: the CFD models under-predicted the discharge by up to 20%. Average error for all models and runs was less than ±8% (Chanel and Doering 2008).

- A series of 2-D CFD models of partial sluice gate openings were developed to compare computed discharge coefficients to experimental measurements and analytical solutions. Good agreement was observed between the numerical model results and results from three experimental studies (Kim 2007).
- A series of simulations compared a 3-D numerical model of two full bays and two half-bays to results from a 1:50 scale physical model of the proposed Keeyask Station on the Nelson River in Manitoba. Model agreement was very close for both full and partial gate openings, with maximum reported disagreement of 5.25% (Teklemariam et al. 2008, Sydor and Waterman 2010).
- A 3-D numeric model of a moving bottom gate at Aswan Dam, Egypt was developed alongside a 1:22 scale physical model to determine the discharge curve under various gate-opening scenarios, with transient effects due to the gate motion included in the numerical model by modeling the gate as a rigid-body 'general moving object' (GMO). The numerical model accurately reproduced experimental water surface profiles and discharges for both pressurized (upstream) and free-surface flow cases, with error magnitude ranging between 0.25% and 3.5% for water-surface profiles and between 1.4% and 4.6% for discharge flow rates. Upscaling the physical model results by Froude number and running the numerical simulations using prototype dimensions reduced the error to between 0.6% and 2.4%. Using 2nd-order, monotonicity-preserving momentum advection instead of the default 1st-order improved free surface elevation errors by 0.5%, which was not deemed sufficient to justify the additional computing resources. Streamwise velocity profiles were consistently over-predicted by the CFD model by 2.6% (Dargahi 2010).
- A 3-D numerical model was developed to address design flaws in the gated and side-channel spillways of the Karian Dam on the Ciberang River, Indonesia. Comparison with hydraulic scale model results showed excellent agreement. The velocity field of the numerical model was used to determine that the flow was disrupted during the PMF event by an inadequately high guide wall in the approach channel. Adjusting the wall upward resolved the flow instability in both the physical and numerical models (Kim et al. 2010).
- 3-D models of the Soyang Dam reservoir, gated spillways, and discharge channels were developed to study flow patterns during the design flood. Comparisons to sparse physical model data demonstrated a maximum 4% difference between physically and numerically modeled discharges. The numerical study also indicated uneven flow distribution through the gates due to a complex flow pattern upstream of the weir (Yoo et al. 2002).

- Simulations of labyrinth spillways:
  - A comparison between a 3-D numerical model, a 1:15 Froude-scale model, and theoretical equations for a redesigned labyrinth spillway on the Dog River, Douglas County, GA showed better agreement between the physical model and numerical model (less than ±10% error for 10 of 11 modeled heads) than between the physical model and theoretical equation (greater than 10% error for 10 of 13 modeled heads, maximum error about ±25%). The study suggests that the theoretical relationships for labyrinth discharge are not applicable to relatively high weirs (Savage et al. 2004).
  - A comparison between 3-D numerical models and physical model results for the final design

of the Hyrum Dam auxiliary labyrinth spillway in Utah, where the labyrinth projects into the reservoir with a curved channel entrance, showed the numerical model results within ±5% of the physical model results for the four cases studied. The models included two cycle-width-to-pier-height (W/P) aspect ratios, including a ratio less than the limit-of-applicability of the design equations. (Paxson and Savage 2006).
  - A similar comparison between a 3-D numerical model and a physical model of the Dog River Dam in the U.S. state of Georgia, where the aspect ratio of the labyrinth is less than the minimum used in the empirical equations, also showed agreement within 5% between physical and numerical models, and illustrated the limitations of the design equations for aspect ratio W/P less than 2.0 (Paxson and Savage 2006).
- Simulations of stepped spillways:
  - A series of 2-D numerical models of a non-aerated, steep, stepped spillway were developed to compare to physical model data and previously published results for similar cases. The numerical models showed good agreement with experimental results, with average discharge difference of 2.4% and maximum discharge difference of 5.0%. Free surface profile differences were less than 9%. Velocity profiles averaged 7% difference and maximum difference 10%. Boundary layer thicknesses were in good agreement with experimental measurements. Turbulent kinetic energy results from the numerical models were in good agreement with published studies of similar problems. The standard κ-ε turbulence model was found to work as well as the RNG κ-ε model (Bombardelli et al. 2010, Meireles et al. 2010).
  - Physical and numerical models were developed for the spillway at Langsett Reservoir, South Yorkshire, England. The crest of the dam carries the public road, which crosses the spillway on a masonry bridge. The spillway is a broad-crested weir followed by a stepped bywash channel. 3-D numerical model results were compared to results from a 1:35 scale physical model. Surface roughness was varied to calibrate the numerical model, and agreement between simulation and physical experiment was excellent (Lavedrine and Woolf 2001).
- Simulations of tunnels, pipes, and draft tubes:
  - One of the earliest applications of FLOW-3D to tube flow was a theoretical demonstration of flow in a water turbine passage, using a rotating reference frame and a fixed blade to create the effect of a rotating turbine (Sicilian 1987).
  - A comparison of a 3-D numerical model and field measurements at the Robert Bourassa dam on the La Grande River, Quebec was developed. The model studied water levels and discharge rates for 7 operating configurations of 8 draft tubes, which are connected to 2 tailrace tunnels by a surge chamber. Observed discharges ranged from 705 m^3/s to 2,319 m^3/s. The difference between simulated and observed discharges was less than ±1.5%, with the average of all error magnitudes less than ±0.75%. The difference between simulated and observed water levels in the surge chamber was less than ±60 cm (1.75%), with average difference magnitude ±20 cm, or 0.57% (Fuamba et al. 2006).
  - A comparison of a 3-D numeric model with a scale physical model of the spillway and four discharge tunnels at Dam Sysenvatn in Norway was developed to examine flow rates with pressurized flow at high heads and free-surface flow at low heads. The geometry was complex, with a submerged discharge pool, partially submerged tunnels, and overtopping bays at high flows. The numerical model showed excellent agreement with the physical model: $R^2 = 0.97$ (Jacobsen and Olsen 2010).
  - Two 3-D numerical models were developed to identify and remedy the cause of parasitic head loss and vortex shedding in the circulating water system at the Texas Utilities (TU) Electric (now Luminant) Martin Lake Plant, a coal-fired generating station. The numerical

model represented the as-built inlet pipe, block, condenser feed pipes, and waterboxes with tubesheets. Model results accurately predicted cyclonic flows, flow transitions, and observed head loss. The second model was used to evaluate remediation, and guided the removal of three guide pylons that eliminated the excess head loss (Williams et al. 1999).

- A series of 3-D numerical models were developed to compare computed drag coefficients of mechanical tube cleaners to bench-scale experimental data and to model the flow within large-diameter tubes and waterboxes with mechanical tube cleaners present. The numerically-calculated drag coefficients showed excellent agreement with experimental measurements, and operational changes made to the distribution density of cleaners based on the model results showed improvement in condenser cleanliness at a two-unit coal plant on the west coast of Florida (Echols et al. 1998).
- A series of 3-D numerical models were developed using FLOW-3D, CFX-TASCflow, and CFX-4 software to design and optimize the Pelton turbines at the Bieudron hydro plant, which at the time of writing were the most powerful, and highest net head, Peltons in existence. CFX-4 provided better velocity profiles of the nozzle-jet flow, while FLOW-3D provided more precise location of the free surface, when compared to experimental test data. FLOW-3D was used alone for simulations of the moving jet-bucket flow in a rotating reference frame. The results of the jet-bucket flow simulations appeared promising, but were difficult to compare to experimental data, leading the authors to conclude that further validation was needed (Sick et al. 2001).

<u>Nappe impingement analysis</u>

- The earliest reported FLOW-3D nappe impingement study was a 2-D examination of transient effects (turbulence, foaming, and recirculation zones) downstream of a drop structure. Alterations in the downstream geometry were demonstrated as solutions to a) partially deflect, b) partially trap, and c) completely contain the submerged jet to minimize recirculation (Hirt 1994).
- A validation comparing a numeric model with measured flow parameters over a drop structure for 0.55 m^3/s (Cook and Richmond 2001) gave good agreement, with the worst prediction being the depth (disagreement of 0.03 m) and velocity (disagreement of 0.2 m/s) at the drop brink.
- A comparison of two numeric models to 1:30 scale models of 5300 cfs skimming and cantilevered outfalls showed good agreement for the jet surface profiles but gave poor agreement for the downstream velocity profiles when compared to time-averaged physical results. RMS/mean disagreement in velocity magnitudes were reported between 7% and 87% for 10 time-averaged mean longitudinal velocity profiles. The observed differences were mainly caused by differences in jet angles, and this could have been caused by the lack of an air entrainment model, the highly transient nature of the test case, scale-of-sampling-volume difficulties in the physical model, or numerical diffusion due to a non-cubic mesh (Cook and Richmond 2001).
- A comparison of a 3-D numeric model against a scale physical model of Soyang dam reported good agreement for the point of nappe impingement and attachment length. The study did not report numerical correlation of the data (Yoo et al. 2002).
- A 3-D numeric model of the Roanoake River Smith Mountain Dam was developed to determine the effect of increasing the maximum PMF. The model was validated against results of scale physical model testing from 1960. The model predicts jet trajectory, nappe impingement point, and spillway performance under flow rates approximately double the original design (Gessler 2005). The cost of the validated numeric model was approximately 20% of an equivalent physical model (Gessler and Rasmussen 2005).

- A series of numerical models and a 1:25 scale physical model were developed to improve jet dispersion and reduce momentum at the impingement to reduce total dissolved gas entrainment at the Boundary Dam sluices and spillway on the Pend Orielle River in Washington. The published project description reports qualitative agreement with photos of the physical model operation (Sweeney et al. 2009).

<u>Pressure/shear distribution and tractive force analysis</u>

- A comparison of a 2-D numerical model to a laboratory physical model of an ogee-crested spillway yielded 94% of 84 pressure measurements that agreed within 5% of model results (Savage and Johnson 2001).
- A comparison of a 2-D numerical model to a US Army Corps of Engineers (USACE) Water Experiment Station (WES) physical model of an ogee spillway showed "excellent" agreement between spillway pressure profiles per the authors (Ho et al. 2001). This study did not include a turbulence model, which may have caused the velocity to be overestimated and thus the pressure to be underestimated in the numeric model.
- A comparison of a 3-D numerical model to a physical model of a concrete gravity dam central spillway in New South Wales, Australia showed "relatively good agreement" between spillway pressure profiles, per the study authors. This study did not include a turbulence model, which may have caused the velocity to be overestimated and thus the pressure to be underestimated in the numeric model (Ho et al. 2001, 2003a).
- A series of 3-D numerical models were developed to analyze intake design alternatives at the Wuskwatim Dam project on the Burntwood River, Manitoba. The simulated scenarios included models to study tractive forces on dumped rock fill during river closure via advancing cofferdams (Teklemariam et al. 2002).
- A comparison of a 3-D numeric model to a 1:36 scale model of a stilling basin with baffle blocks showed agreement between numeric and physical model pressure profiles within the accuracy of the piezometers (Cook et al. 2002),
- A comparison of a 3-D numerical model to a 1:50 scale model was developed for the Hume Dam Spillway in NSW. Pressure heads agreed well along the centerline of the spillway, while the numeric model under-predicted the pressure near the edge of a pier while capturing the trend (Ho et al. 2003b).
- A study of the effect of interim raising of the dam crest at Warragamba Dam, NSW, was conducted to determine if the excess negative pressure created over the crest would be sufficient to lift the drum gate out of its chamber. Results were favorably compared to those of a 1:100 scale physical model. Loads on the gate structure were quantified and possible crest modifications to reduce the suction were simulated. Drag and lift forces were also determined for radial gate arms and trunnions submerged during the revised Probable Maximum Flood (PMF) event using the CFD model results combined with a structural finite-element (FE) analysis (Ho et al. 2003b, 2006).
- Five studies were performed for the Hume (New South Wales), Buffalo (Victoria), Wivenhoe (Queensland), and proposed Tallowa (New South Wales) dams as well as the Goulburn weir (Victoria). 3-D numerical model results were compared to results of physical models ranging in scale from 1:50 to 1:100. The pressure distribution was used with the self-weight of the structure and estimates of uplift pressure to determine the factor-of-safety against overturning and sliding. Model results were also applied to FE analysis to determine the potential for structural deformation of raised gates, and to examine the effectiveness of proposed solutions (Ho et al. 2006).

- A series of 2-D submerged ogee spillway models were compared to physical model pressure tap measurements showed: for a flip-bucket spillway, absolute pressure differences of less than ±1 cm for 133 (91.7%) of 145 pressure measurements and an absolute maximum difference of less than ±2 cm. For a horizontal-apron spillway, the models gave absolute pressure differences of less than ±1 cm for 106 (94.6%) of 112 pressure measurements and an absolute maximum difference of less than ±2.5 cm (Johnson and Savage 2006, Savage and Johnson 2009).
- A 2-D numerical sectional model of the Wanapum Spillway on the Columbia River, Washington, was developed to analyze the contribution of hydrodynamic forces to the stability of the spillway monoliths during the probable maximum flood (PMF) of 1.4 million cfs. Pressure results showed good agreement with physical model studies. The CFD model results indicated that the conventional stability analysis methods used to design the dam underestimated the sliding safety factor of the monolith by 40% to 50% (Griffith et al. 2007).
- A series of 3-D numerical models of the Cabinet Gorge Dam were developed to analyze pressure distribution, variation, and magnitude within the tunnels. The model was validated against a physical scale model study, and showed "reasonable" agreement for average pressure and pressure distribution for all discharges modeled. The variation in pressure due to turbulence was smaller in the numerical model (±3 ft) than in the physical model (± 11 ft), and the physical model was deemed more accurate based on the small length and time scales of the turbulent eddies (Groeneveld et al. 2007).
- A series of 2-D CFD models of partial sluice gate openings were developed to compare computed pressure profiles along the channel bottom and on the partially-open gate against experimental lab measurements. Good agreement was observed between the numerical model results and experimental results from two research teams (Kim 2007).
- A 3-D numeric model of a moving bottom gate at Aswan Dam, Egypt was developed alongside a 1:22 scale physical model to determine the discharge curve under various gate-opening scenarios, with transient effects due to the gate motion included in the numerical model by modeling the gate as a rigid-body 'general moving object' (GMO). The numerical model accurately reproduced experimental pressure distributions for the free-surface flow cases, with some divergence from experimental values at the extremes of high- and low-head. The pressurized flow cases consistently over-predicted the experimental pressures with a maximum of 10% difference (Dargahi 2010).
- A numerical model studied pressure fluctuations caused by increasing the design head by 100% at the Boundary Dam spillway on the Pend Orielle river in Washington. The model examined the potential for vibrations to develop at the crest, though no comparison to physical data was reported (McPhail et al. 2010).

<u>Tailrace and stilling basin analysis</u>

- A 2-D validation case simulated flow characteristics of hydraulic jumps. Upstream and downstream free-surface elevations were within 0.01 m of experimental data (Cook and Richmond 2001).
- A 3-D validation of simulated velocity profiles for a 1:40 scale model of a stilling basin with baffle blocks: 73% of velocity samples were within ±1σ (σ = standard deviation) and 95.5% within ±2σ of the physical model data (Cook et al. 2002).
- A comparison of a 3-D numeric model with a 1:80 scale physical model of a tailrace and stilling basin showed entrained lateral flows that were observed in the physical model with average velocity difference of 1.4 ft/s and R^2 = 0.81 (Cook et al. 2002).
- A qualitative comparison of recirculation zones for 5,700 cfs in a stilling basin with deflector at

the John Day dam (Cook and Richmond 2001) visually matched photographic data, with the notable absence of fluid 'bulking' due to air entrainment (the FLOW-3D air entrainment physics model was not included in the software at that time).

- A comparison of a 3-D numeric model to a 1:30 scale model of a 5,300 cfs skimming outfall gave root mean square (RMS) errors less than 25% for 4 of 5 time-averaged mean longitudinal velocity profiles, with average of all RMS errors 24% (Cook and Richmond 2001).
- A comparison of a 3-D numeric model to GPS-tracked drogue motion at the Bonneville Second Powerhouse with measured average flow of 2,500 cfs showed good agreement between the simulated velocity field and the entrainment of a GPS-tracked drogue in a surface gyre (Cook and Richmond 2001).
- A comparison of an uncalibrated 3-D numeric model to acoustic Doppler current profiler (ADCP) velocity measurements of the Bonneville Second Powerhouse tailrace with average measured discharge of 141,000 cfs found 46% of velocity samples were within ±1σ and 99% within ±2σ of the measured values. The average of all RMS errors was less than 20% (Cook and Richmond 2001).
- A comparison of a 2-D numeric model with both a 1:100 scale physical model and in-situ prototype measurements of the Porto Colombia Power Plant dam (Rio Grande, Brazil) included the Creager-profile spillway (ignoring the effect of the radial gates), intake, tailrace and stilling basin with associated hydraulic jump. The floor pressure distribution of the numerical model was in good agreement with the scale model and in-situ measurements, and the 2-D model reproduced observed recirculation patterns and jump location as a function of discharge (Amorim et al. 2004).
- A 3-D model of the Wivenhoe, Queensland dam spillway and stilling basin predicted the hydraulic jump there and examined recirculation patterns in the stilling pool and basin integrity as a function of the pressure distribution on the basin floor (Ho et al. 2006).
- A 3-D model of the Blowering, New South Wales long spillway chute downstream of the ogee crest predicted diagonal shockwaves during the supercritical flow of the revised PFM. The predictions were checked against validations with known solutions proposed by Chow (1959) and the USACE (1994). The simulated flow elevations were used to determine how far the chute walls needed to be raised to contain the PMF discharge (Ho et al. 2005, 2006).
- A 3-D model of the Robert Bourassa dam on the La Grande River, Quebec, was developed to examine pressure wave effects in a surge chamber connecting 8 draft tubes to 2 tailrace tunnels. The model examined 7 operating configurations of the draft tubes and compared simulated discharges and water levels to observed data. The difference between simulated and observed discharges was less than ±1.5%, with the average of all the magnitudes less than ±0.75%. The difference between simulated and observed water levels in the surge chamber was less than ±60 cm (1.75%), with average difference magnitude of ±20 cm, or 0.57% (Fuamba et al. 2006).
- Two 2-D numerical models were developed to analyze the predictions of the internal flow structure in submerged hydraulic jumps. The RNG and standard κ-ε turbulence models were compared and the RNG model was found to perform better near the floor (Raiford and Khan 2006).

<u>River reach and intake analysis</u>

- A 3-D numerical model of flow around a pier at the Akashi Kaikyo Ohashi bridge was compared to field measurements. The study compared occurrence periods for shed vortex and Von Karman vortices when the Reynolds number was transcritical. The numerical simulation gave results in good agreement with the field measurements (Ishino et al. 1993).

- A 3-D numerical model of flow in a trapezoidal channel compared fluid surface height distribution to 1:20 scale model experimental data. Surfaces were modeled as hydraulically smooth. Numerical results agreed with experimental data (Miyamoto and Ishino 1995).
- A series of 3-D models were created to evaluate the hydraulics of leaving or removing piers in the water passage of a proposed powerhouse intake at Whitesand Dam on Reindeer Lake, Saskatchewan. The simulation results included surface elevation, flow velocity and direction, and pressure head. The analysis demonstrated that the piers cause flow patterns with unnecessary head loss, and removing them was demonstrated to save 8,640 megawatt-hours per year by improving approach conditions to the powerhouse (Groenveld 1999).
- A 2-D numerical dam break simulation was performed for the North Dike of the Grand Rapids Station on the Saskatchewan River, Manitoba. Results of the model were considered superior to 1-D DAMBRK simulations due to complex and steep topography. Results were used to generate improved inundation maps which included areas that had previously been omitted using 1-D results (Groenveld 1999, Teklemariam and Groenveld 2000).
- A series of 3-D numerical models of the power intakes at the Kelsey generating station on the Upper Nelson River in Manitoba compared simulated velocity profiles, surface elevations, and flow features to observed conditions and physical models. Three scenarios and a calibration of the power intakes were conducted using CFD models to improve turbine efficiency at the power intake. The calibration visually compared the simulated flow field to historical field tests, and was considered successful, in part because the numerical results accurately predicted a vortex location. The scenarios included various opening configurations of the seven gates. Intake velocity distribution results were provided to the turbine manufacturer for analysis and review (Groenveld 1999, Teklemariam and Groenveld 2000, Teklemariam et al. 2002, Wang and Sydor 2008, 2010).
- A 3-D numerical simulation was developed to study flow structures at the Conawapa Stage II Diversion on the Nelson River, Manitoba. Results were compared to a physical scale model. The numerical results closely reproduced the flow patterns and velocity magnitudes measured in the physical study (Teklemariam and Groenveld 2000, Teklemariam et al. 2002).
- A customization of FLOW-3D was used to develop a novel mathematical model of fish behavior in complex 3-D flow fields. Japanese char were modeled as 'adaptive particles', using a modification of FLOW-3D's Lagrangian particle model to include adjustments of and for speed, velocity vector, territorial factors, reproduction, and death by predation and age. Physical experiments in flumes using live char gave adequate agreement with the numerical model (Kitamura et al. 2001).
- A series of 3-D numerical models were developed to analyze intake design alternatives at the Wuskwatim Dam project on the Burntwood River, Manitoba. The simulated scenarios included models to study proposed designs to minimize cross-flow velocities and uneven flow distributions at the intakes, to induce the early formation of a stable ice cover, and to simulate flow profiles, velocities, and tractive forces on dumped rock fill during river closure via advancing cofferdams. The model results closely matched rating curves measured upstream of the Taskinigup Falls (Teklemariam et al. 2002).
- A comparison of a 3-D numerical model with measured ADV velocity profiles for a highly sinuous reach of the Embarras River in Illinois showed that the numerical model accurately predicted the main features of the velocity field, although the RNG turbulence model under-predicted some secondary circulation patterns (Rodriguez 2004).
- A 3-D numerical model was developed to predict thermal mixing and stratification (density currents) at the confluence of the Snake and Clearwater rivers near the Idaho/Washington

border. Comparison of field-measured surface temperatures showed thermal agreement within 1° C, and the model was further validated against satellite imagery of surface plunge lines (Cook and Richmond 2004).

- A 3-D numerical model was developed to simulate mixing and dispersal of a thermal plume near a cooling water intake structure at the Connecticut Yankee Nuclear Power Plant on the Connecticut River. The study identified the extents of the hydraulic zone of influence (HZI) during low flows, under tidal influence, and included the effects of the dispersing thermal plume. Passive organisms (representing fish eggs and fish larvae, among other life forms) were modeled as scalar concentrations released continuously from 19 locations on the reach in order to determine the extents of the HZI. The size, shape, and location of the thermal plume agreed closely with infrared aerial photographs. Ecological results (the location and extent of the HZI) compared well with ecological sampling and entrainment study data (Richardson and Dixon 2004).
- A series of 3-D models of river reaches with Kinoshita-generated meandering bends were developed to study the velocity distribution in the reaches. Results confirmed previous studies that had shown that the highest velocities are concentrated near the inner bank for channels with low to mild sinuosity, and the outer bank for channels with high sinuosity, and showed good agreement for locations of recirculation cells and regions of reduced turbulent activity (Abad and Garcia 2005).
- Three 3-D numerical models of the Knoxville, TN, Beaver Creek Research Station (an urban, incised, channelized stream) were developed to characterize turbulent hydraulic structures during bankfull discharge of 1.68 m^3/s. The scenarios were of the channel with all existing trees present, with all trees removed, and with a restoration design keeping only three stands of trees. The results indicated the presence of small circulation cells scaled to the degree of turbulence caused by the trees, as well as cross-channel flow patterns in the restoration design channel which reduce the energy available for further incising the channel (Dworak 2005).
- A series of 3-D numerical models were developed to compare FLOW-3D's ability to model porous media to physical model measurements of sweeping and approach velocities at varying flow rates near perforated and wedged fish screen intakes. Two physical models were used: one with a 10:1 ratio of approach to sweep velocity, and one with a 20:1 ratio. The 10:1-ratio model gave more linear results. For both models, the numerical simulations reproduced the approach and sweep velocities measured by Acoustic Doppler Velocimeter (ADV) with very good agreement: less than 6.1% relative error magnitude was observed for the majority of measurements (Ho et al. 2006, 2011).
- A series of 3-D numerical models were developed to analyze the current flow conditions of a power canal and proposed alterations to improve fish bypass efficiency at the Grand Falls Power Canal Fish Bypass Diversion System. Four numeric models simulated current conditions for total flows ranging from 211 to 326 m^3/s. Numerical results predicted observed and measured phenomena, including pulsing and spatially-varying velocity distributions, rotating currents (boils), eddies, and regions of reduced velocity (Woolgar and Eddy 2006)
- A series of 3-D numerical models were developed to analyze the effect of varying releases on the velocity profile of a natural series of cascades downstream from the Link River Dam in Klamath Falls, Oregon. 1,000, 2,000, 3,000, and 4,000 cfs discharges were simulated. Velocity profiles indicated that the probability of fish passage through the cascade decreased as flow increased, identified the likeliest route of passage, and identified the likeliest point at which passage becomes impossible during the flows modeled (Mefford and Higgs 2006).
- Pre- and post-design 3-D numerical models were developed to predict the performance of a

proposed fish ladder restoration at Third Dam on Somes Brook, Maine. The pre-design model accurately predicted the observed flow patterns through the deteriorated ladder, and the post-design model surface profiles matched post-construction photos (Richardson 2007).

- 3-D numerical models were developed to analyze errors in Acoustic Doppler Current Profiler (ADCP) discharge measurements caused by flow-field perturbations induced by the ADCP apparatus itself. Simulation results were compared to field measurements, and the profiles showed excellent agreement with mean differences of 0.1%, coefficient of variation of 0.2%, and 1% maximum deviation from observed data at any point. The field and numerical results indicated that the ADCP-induced perturbations cause errors in measurement that are correlate to the ADCP-Reynolds and Froude numbers (Mueller and Wagner 2007).
- A series of 3-D numerical models were validated against data from a physical flume study of channel flow around and through a groyne. The physical model measured tip velocity, separation length, and velocity vector angles. Four groyne-length-to-channel-width ratios, five groyne porosities, and a variety of flow rates were modeled. The computed velocities and separation lengths varied from the physically measured data by less than ±5% for groyne porosities up to 40% (Ho et al. 2007).
- A series of 3-D numerical models were used to study the flow field effect of five bendway weirs and two additional upstream weirs along a meander bend of Sugar Creek in Illinois. The first 3-D model was validated against field data collected during a period of low flow (1.41 m^3/s), and then the validated model was used to predict the flow field during medium- and high-stage flows. Comparisons of field data and numerical model results showed good visual agreement of velocity profiles, though the numerical model consistently over-predicted velocity in the weirs, at most points by less than 0.2 m/s. The discrepancy was believed due to the model's use of constant surface roughness for both banks and channels, as well as limitations in the RNG turbulence model that do not capture small-scale internal vortex-shedding from the weir tips (Abad et al. 2008).
- A comparison of numerical simulations to 1:20/1:13 scale physical model results and empirical equations was conducted to determine risk factors for direct inflow and sidewise breach discharges into underground facilities during flooding events. Good conformity was observed between results of numerical simulations and laboratory data for sidewise breach discharges (Oertel and Schlenkhoff 2008).
- A series of 3-D numerical models were developed to analyze and refine pre-commitment and post-construction designs of the proposed Keeyask Station on the Nelson River in Manitoba. The models were used to refine the designs of proposed cofferdam layouts, the powerhouse intake, channels, and the river closure and management strategies. The spillways, forebay, and tailrace were also designed with assistance from the numerical models. The models were calibrated against historically-measured stage-discharge curves with maximum difference of 5.25%, and the topography was updated during the calibration process using historical photos of the river bed during low-water events. The CFD model was used to lower the approach channel entrance elevation and reduce bank excavation costs (Teklemariam et al. 2008, Sydor and Waterman 2010).
- A study of the suitability of using FLOW-3D for dam break simulations found that the software could model dam breaks under realistic conditions using reasonable computational resources. After comparing several physical test cases to FLOW-3D and another software package (RIVER2D) it was determined that FLOW-3D provides considerably more accurate results. A qualitative test dam break simulation was successful but suggested that dam break models using FLOW-3D will be limited by computational resources to the regions relatively close to the dam

(Vasquez and Roncal 2009).

- A 3-D "far field" numerical model of downstream water levels for flow events through existing and proposed floodway outlet structures at the Red River Floodway that protects the City of Winnipeg (Manitoba) from flood damage. The numerical model was calibrated against photographic, anecdotal, and physical model observations with good agreement. The model was set up to compare results to existing HEC-RAS models. The proposed outlet structure design used for the predictive part of the model was designed using FLOW-3D as well, and intended to pass a 130% increase in design flow (Gendzelevich et al. 2009).
- A 3-D numerical model was developed and calibrated to study the flow field in and around a sturgeon spawning ground downstream of the Pointe Du Bois generating station on the Winnipeg River in Manitoba as part of a comprehensive fish habitat study. The numerical model compared favorably against ADCP velocity measurements taken during field studies over three consecutive years and against aerial photography showing flow patterns over the spillway rapids (Brown et al. 2009).
- A 3-D numerical model was developed to compare the three-dimensional flow field velocity distribution and the bed shear stress distribution to ADCP measurements at three cross sections of a regulated reach of river near Trondheim, Norway. Results using the default 1^{st}-order (upwind differencing) momentum advection option showed more numerical diffusion. Good agreement between field measurements and the numerical model were achieved using higher-order momentum advection (Rüther et al. 2010).
- A 3-D numerical model was developed to compare free surface profiles and afflux near a series of bridge piers to experimental measurements. The upstream and downstream water surface elevations showed good agreement with the experimental data, and afflux values averaged 5% agreement, with maximum disagreement of 14.5% and minimum disagreement of 0.02% (Kocaman et al. 2010).
- A comparison of 3-D and "shallow water" numerical models to laboratory experiments simulating flow after a dam break found good agreement of the time-evolution of water surface profiles. The full 3-D model performed better than the "shallow water" approach, but the authors found the latter gave acceptable results at a much reduced computational expense (Ozmen-Cagatay and Kocaman 2010).
- A comparison of a 3-D CFD model of saline stratification and wind and current mixing in the Russian River estuary, California, showed good qualitative agreement with field observations of density contours (Sahoo et al. 2010).
- A 3-D numerical model and physical model were developed to study boulder-roughened ramps and slides for river rehabilitation and fish passage. The numerical model showed good qualitative and quantitative agreement with the physical model. Drag coefficients for the boulders were developed, and ranged from 0.8 to 6.5, suggesting that the typical literature value of 1.5 may be inappropriate in complex flow fields (Oertel et al. 2010).
- A series of 3-D numerical models were developed along with laboratory experiments to compare the prediction of drag coefficients for flooded, built-up areas. The numerical results showed very good agreement with the experimental drag coefficient, which was observed to vary with the 2/3 power of water depth (Choi et al. 2009).
- A series of 3-D numerical models were developed to examine flow interactions between a sluiceway, intake structure, and rockfill dam at Ashlu Creek Hydroelectric Plant in British Columbia. The project design flow is 29 m^3/s. The CFD simulations indicated that a deep, fast vortex could be expected. Additional models examined groin alternatives to remediate the flow pattern, ultimately identifying the role of a submerged concrete retaining wall in causing the

vortex formation. The final CFD-modeled design indicated a low-cost solution that greatly reduced sluiceway entrance velocities and the associated likelihood of vortex formation (Briand et al. 2010).

LESSONS LEARNED AND GUIDELINES FOR MODELING DAMS AND RIVERS

There are several points that can be gleaned from the above studies. First, it is clear that in a majority of cases, the TruVOF®/Navier Stokes method provides reasonable to excellent agreement with physical experiments. This in turn implies that the solution method is robust enough to "work" for a wide variety of flow geometries, mesh resolutions, simulation durations, and optional physical models. This robust nature is encouraging for modelers (e.g. Cook and Richmond 2001, Teklemariam et al. 2002, Rodriguez et al. 2004).

Limitations and options for turbulence models

There are, however, some limitations that become apparent. One is the under-prediction of secondary circulation patterns and eddy effects when using the standard κ-ε and RNG κ-ε turbulence models. The κ-ε models are examples of *two-equation isotropic Reynolds-Averaged Navier Stokes (RANS) closures*. The RANS turbulence closures are limited in their ability to resolve highly swirled flows because they assume a linear relationship between fluid stress and strain rate and are therefore insensitive to normal stresses and anisotropy; this causes these models to damp vortices and secondary flows (e.g. Lyn 2008). This limitation is well documented (e.g. Rodriguez et al. 2004, Abad et al. 2008), and is not limited to FLOW-3D (e.g. Lyn 2008, Rodi 2010). There are several examples of successfully modeling vortex formation listed in the case studies above using RANS turbulence models (e.g. Briand et al. 2010, Wang and Sydor 2010). As a rule of thumb, where a physical model will exhibit a fully-formed air core, FLOW-3D will exhibit a strong surface depression instead, and knowing that the CFD model using a RANS turbulent closure will damp the vortex formation is important in interpreting results.

A promising alternative to RANS turbulent closures are *large eddy simulation (LES)* turbulence models. LES models greatly reduce the empiricism in modeling turbulence, and are clearly superior in modeling turbulent eddies and vortices (Smith 2007, Rodi 2010). The increased accuracy comes with a cost: LES models require mesh resolutions small enough to resolve individual eddies, and thus they are extremely computationally intensive and usually require very long run times, which are generally prohibitive for industrial-scale problems (e.g. Lyn 2008). LES models were originally developed for atmospheric research, and have required improvement to remain accurate and stable in the presence of solid surfaces such as channel beds and spillways (Rodi 2010).

LES modeling is becoming more feasible for dam-scale problems as computer hardware increases in power, extra computer memory becomes less expensive, and parallelization of software becomes more efficient. LES models are particularly feasible when more than one computer is applied to the solution (Rodi 2010). This distributed computing approach uses *message passing interface* (MPI) architecture, and currently requires a relatively advanced understanding of operating systems to implement. Flow Science and other CFD software publishers have developed MPI versions of their software, although the author's personal experience and a cursory literature review do not yet indicate widespread use of MPI in river and dam applications.

Some situations aren't feasible to model with FLOW-3D

Another important factor in numerical simulations is the ability of the software to complete a simulation in a reasonable amount of time, reasonable being defined as within a project phase deadline. In academic settings the deadline might be quite long, but for many design and engineering firms the time spent to complete a project has significant financial implications. The value of time when running CFD models limits the software to certain classes of problems, namely the types described above.

Sediment scour problems are generally limited to event- and reach-level simulations. Multi-year and basin-wide sedimentation problems are currently beyond the capabilities of the software to model in a reasonable period of time, primarily because the solver makes no assumptions that the flow is steady-state, or amenable to depth averaging, or hydraulically stable, and the governing equations are solved in three dimensions. Solving these governing equations requires a variable time-step on the order of a second or two to less than a millisecond, it is immensely computationally intensive to model scenarios lasting more than a few hours of real-world time, and 2-D or 1-D steady-state or quasi-steady-state software packages should be considered until FLOW-3D includes coupling between its 'shallow water' model (depth-averaged gradually-varied flow assumption) and sediment transport/scour models.

Similarly, very high speed flows with significant fluid breakup such as those found in spray nozzles are time-consuming to model with FLOW-3D. The small time step required to resolve the transport of hundreds or thousands of individual droplets on a very fine mesh is often time-prohibitive. Coarser simulations of composite fluid effects in jets and sprays have been successful with FLOW-3D (e.g. Sweeney et al. 2009), but for detailed analysis of high-pressure sprays (such as those occurring in turbine units), CFD packages using body-fitted meshes are often more accurate (e.g. Sick et al. 2001).

Very thin curved geometries such as turbine blades are also difficult to model because of the orthogonal mesh used by the TruVOF approach. Resolving thin blades requires a very fine grid, which increases the run time of the simulation dramatically. In addition, the near-wall approximations for velocity that the turbulent closure models apply assume fully-developed isotropic turbulent flow. For rapidly rotating blades this assumption leads to inaccuracies, and so most CFD modeling of turbine blade rotation utilizes CFD packages with body-fitted meshes (e.g. Sick et al. 2001).

The importance of verifying results

Another important aspect of modeling is that CFD results typically look superficially accurate. The detailed three-dimensional, color-coded output is perfect for reports, presentations, and publications, and can lead to a false confidence in the results. While numerical model results usually trend accurately, they do not always fully capture the detail of the physical case they are modeling, and they sometimes markedly over-predict or under-predict reality (e.g. Cook and Richmond 2001). It is for this reason that successful models require some degree of error-checking. Good practice includes back-of-envelope checks of numerical output, comparison to photographs, or comparison to studies of similar cases (e.g. Teklemariam et al. 2002, Gessler 2005) When the results need to be highly accurate for safety or economic reasons, a physical model may be required. In these cases, CFD simulations can guide the implementation of the physical model and thus lower the final costs of the project (e.g. Gessler and Rasmussen 2005).

Minimizing numerical error

A common cause of model error is *numerical error*. This is an unphysical loss of accuracy due to the way the CFD package solves and governing equations, and is especially dangerous near mesh boundaries. Numerical errors that reduce the result accuracy without crashing the simulation can be introduced by the following sources (Hirsch 2007):

- the discretization of space and time,
- unconverged implicit solutions, especially pressure,
- loss of precision due to rounding,

- inaccurate or inexact topography and geometry,
- incorrect or incompatible boundary conditions,
- user error in solver parameters such as relaxation and iteration coefficients,
- simplifications and approximations in models, especially relating to turbulence closures and multiphase flows, and
- bugs (errors) in the software code itself.

Numerical error is a potential problem with all 3-D software packages (Hirsch 2007), but there are ways to avoid it. Regardless of the code, a major source of numerical error is the quality of the mesh. The following rules-of-thumb will help minimize diffusion error (Hirsch 2007):

- Place mesh boundaries, including the boundaries between contiguous mesh blocks, as far as reasonably possible from areas with large gradients, including recirculation zones, hydraulic jumps, and complex geometry.
- Use grid-refinement (called nested blocks in FLOW-3D) wherever large gradients are expected. The regions of large change in flow properties often call for smaller cell sizes so that the solution accuracy is maintained.
- Avoid distorted cell aspect ratios. The most numerically stable cell ratio is the cube, although when the flow direction is mostly in the horizontal then the vertical cell size can be made smaller to better resolve the velocity distribution.
- Avoid discontinuous stretching of the mesh grid. In FLOW-3D, grid stretching is automated so that it is always continuous and therefore always stable. However, the user can force the grid to stretch in order to maintain cell edges at certain locations, and too much specification of these edges can distort the mesh enough to cause a loss of accuracy ranging from negligible to significant.
- Avoid discontinuities between cell sizes at mesh block interfaces. Wherever two mesh blocks adjoin, the maximum ratio between cell face lengths should be 2:1. Also, the cell faces should be exact multiples of each other, and should align perfectly to avoid artificial momentum diffusion caused by interpolation at the boundary.

<u>Software evolution</u>

A final issue that has both positive and problematic connotations is the continual evolution of CFD software. Commercial CFD codes are being improved monthly, with many companies releasing significant updates on a yearly or more frequent basis. While the overall solutions may be approaching greater accuracy over time, an individual simulation run on two different versions of the same code may exhibit noticeably different results.

The numerical difference is often minimal when compared to the scatter of physical data, but not always: an example of this can be found in FLOW-3D's default hiding/exposure function for the multi-species sedimentation model. The implementation of multiple grain species, with a user-customizable hiding/exposure relation, first appeared in FLOW-3D v9.4.0. Brethour and Burnham (2010) describe the model and give the original hiding/exposure function, which is that of Egiazaroff (1965). Subsequent investigation revealed the asymptote and unphysical behavior of the function for species with size less than 40% of the mean. Beginning in FLOW-3D v9.4.5, the model included a correction procedure first suggested by Ashida and Michiue (1972). Results for multi-species simulations between v9.4.2 and v9.4.5 are significantly different. This example serves to illustrate the possibility of inter-version variation with CFD models, especially those that are newly being developed.

CUTTING EDGES IN FLOW-3D MODELING OF DAMS AND RIVERS

The case studies described above provide an overview of some of the most common applications of CFD for dam safety and design. A few of the studies provide interesting glimpses of potential future applications and areas of development. Four development focuses are selected for further discussion: modeling sediment and rock transport and deposition, modeling transient effects due to moving geometry, modeling cavitation formation and damage, and modeling air entrainment. The following sections discuss the ongoing work in these areas, as well as current limitations and suggestions for application of FLOW-3D to these classes of problems. It should be noted that many other CFD software packages also have methods for approaching these problems. A discussion of strengths and limitations of each software package is beyond the scope of any single paper. The discussion below is limited to FLOW-3D.

Modeling sediment and rock transport and deposition with FLOW-3D

Numerical simulation of sedimentation physics is a cutting edge field of research and development for many private and public institutions. There are a multitude of 1-D, 2-D, and 3-D sediment transport models available for fluid flow, many of which are detailed in Thomas and Chang (2008) and Spasojevic and Holly (2008). FLOW-3D has been used to investigate sediment and rock transport since at least 1995 (see case studies below), usually by using velocity and pressure distribution results with empirical equations (e.g. Kent and Richardson 1999, Qiu 2008). There has been an evolving scour model included with FLOW-3D since 2002. The model has been extensively revised to its current form, which allows up to ten unique sediment species, includes functions to describe the effects of hiding and exposure, bed angle and sediment angle-of-repose, suspended load entrainment, advection, and hindered settling, as well as bed load transport. Entrainment, bed-load, angle-of-repose, and hiding/exposure models are user-customizable because the empirical nature of scour modeling prohibits a one-size-fits-all solution. With the advent of the sediment and scour model, modeling of deformable beds became possible, although it is still computationally expensive to model scour phenomena larger than the scales of the reach and event. Summaries of an English-language literature review of FLOW-3D sediment transport and scour models are included below.

- A 3-D numerical model of flow around a cylindrical pier and within the equilibrium scour hole showed good quantitative and qualitative agreement with experimental data. Observed differences in the results were attributed to the selected turbulence model, the time-averaging of the experimental data as opposed to the instantaneous output of the numerical model, and the experimental method of determining bed shear stress (Richardson et al. 1995).
- Three 3-D numerical models were developed of a proposed bridge over the Quinnipiac River in New Haven, Connecticut. The model domain included two central lift piers, two cellular cofferdams, and a continuous timber fender system. The fender was modeled as a porous obstacle with 30% porosity. The simulation scenarios included the as-built configuration, the crossing with the fender removed, and the crossing with the piers only. The 100-year (12,100 cfs) event was taken as the design flow. Downstream eddies were observed in the results, which were applied to the CSU equation to calculate potential scour depth (Kent and Richardson 1999).
- A series of 3-D FLOW-3D numerical models were used in conjunction with 1-D HEC-RAS models to test the hydraulic performance of a proposed pool-riffle design for the West Fork of the North Branch of the Chicago River, Illinois. FLOW-3D was used to test the pool-riffle unit hydraulics, while HEC-RAS was used to test the reach hydraulics. FLOW-3D results demonstrated that the proposed design does not increase the likelihood of bed or bank erosion

(Rodriguez et al. 2001).

- A series of 3-D numerical models were developed to study tractive forces on dumped rock fill during river closure via advancing cofferdams at the Wuskwatim Dam project on the Burntwood River, Manitoba (Teklemariam et al. 2002).
- A series of 3-D numerical models were developed to study FLOW-3D's ability to model vortex shedding and associated scour pit formation near cylindrical pipes in coastal environments with waves. Numerical results using the FLOW-3D sediment scour model showed good agreement with physical experimental results for velocity distribution, vortex structure magnitude, vortex shape, vortex location, and numerical time-scale predictions during the initial stages of scour. Time-scale of scour development was significantly under-predicted for later stages. The shape of the numerical scour hole agreed with experiment, but under-predicted the depth. Two turbulence models were used; the Large Eddy Simulation model (LES model) and the standard κ-ε model. The LES model showed superior prediction of vortex structures (Smith 2007).
- A series of numerical models were developed to investigate the effect of different boundary conditions on flow simulations across low-water stream crossings. The results were found to be accurate, and suitable for use with empirical sediment transport equations (Qiu 2008).
- Qualitative simulations of local scour around ten cylindrical bridge piers in tidal flow were conducted using FLOW-3D's sediment scour physics model. The numerical results reproduced experimental observations of scour. A simplified (3-pier) simulation showed qualitative agreement with measurements reported in the literature for scour under tidal flow reversal (Vasquez and Walsh 2009).
- Four numerical simulations of loose sand erosion caused by a plunging jet from an overtopped floodwall used the FLOW-3D sediment scour physics model. Comparison with lab experiments showed very good agreement, with a maximum observed difference in maximum scour depth of 16% (1.4 cm). Agreement varied for scour hole location and time to reach equilibrium depth, from near identical agreement to up to 50% disagreement: 10.4 cm and 1.5 seconds difference, respectively (Wang et al. 2009).
- FLOW-3D was used in conjunction with physical modeling, finite element method (FEM) analysis, and empirical checks to predict the maximum scour depth around the piers of Incheon Bridge, South Korea (Cho 2010).
- Three numerical test cases were studied to determine the suitability of using the FLOW-3D Lagrangian particle model to determine suspended sediment motion paths. The cases represented a desander, a run-of-river intake, and a reservoir powerhouse intake. Results were qualitative, and conformed to expectation (Vasquez 2010).
- Numerical models of a free-falling jet and associated scour hole were developed using the FLOW-3D scour model. Results were compared to lab experiments and U.S. Federal Highway Administration (FHWA) design equations. The depth, length, and width of the numerical scour hole agreed more closely with the physical experiment than the FHWA equations. The maximum difference between numerical and physical experiment was 5% (Amiraslani et al. 2010).
- Numerical models and physical flume models of pier-scour at Houfeng Bridge, Taiwan, were compared. The numerical model was first validated by comparing the oscillating height of the standing wave on the first pier to the physical experiment: good agreement was observed. The scour pit formation agreed very closely with the physical experiment, although the numerical model eroded more quickly than the physical model, indicating a possible time-scaling effect (Lin et al. 2010).
- Numerical models of sediment transport were compared to results from a 1:35 Froude-scaled

physical model of the Räterichsboden reservoir and tunnel system. The physical and numerical models were then validated against the prototype by turbidity measurements (scattered light photometer and Imhoff cone samples) taken during an annual functional test of the bottom outlet of the reservoir. The three methods showed good agreement, which the authors identified as 'similitude,' keeping in mind that all sediment models, physical as well as numerical, come with a great deal of scatter in results (Möller et al. 2011).

Modeling transient effects due to moving geometry with FLOW-3D

FLOW-3D can model solid-fluid coupling with moving three-dimensional objects. The objects are coupled with fluid flow, subject to pressure, shear, and buoyancy forces, and may be constrained in rotation, constrained in translation (lateral and vertical movement), or a combination. Control forces and torques can be added to simulate the action of hydraulic machinery, and a variety of springs and ropes can be added to further constrain the motion of the object. The general rigid moving object model was added to the software in 2005. Plastic deformation of objects using finite element method and body-fitted meshes was introduced in 2011. While many, if not most, uses of FLOW-3D go unpublished, some studies are presented to the public. Three applications of rigid-body motion with significance to dam and river studies are highlighted below.

- Moving gates can be modeled where the transient effects of the gate are important. One study compared a numeric model of a moving bottom gate at Aswan Dam, Egypt, to a 1:22 scale physical model to determine the discharge curves under various gate-opening scenarios with transient effects due to the gate motion included in the numerical model. Error magnitudes were between 0.25% and 3.5% for water-surface profiles and between 0.6% and 2.4% for discharge flow rates after upscaling the physical model results by Froude number. Streamwise velocity profiles were consistently over-predicted by the CFD model by 2.6%. Free-surface flow pressure distributions were predicted accurately, while pressurized-flow pressure magnitudes were consistently over-predicted by up to 10% (Dargahi 2010).
- Water hammer simulations have been conducted by a number of users of FLOW-3D (e.g. Bouton 1991). Applications in dam and hydroelectric modeling have mostly been focused on simulating hammer in penstocks, discharge tunnels, and intakes due to closure of valves and gates (e.g. Fuamba et al. 2006). Bouton (2006) found that complex interactions between water and air bubbles occur during water hammer events that are not included in purely analytical treatments of water hammer. The use of the GMO model to identify and quantify water hammer in valve designs has been demonstrated (Flow Science 2005a).
- Log and ice-jam studies have been developed using the FLOW-3D general moving object model. One example is a series of seven 3-D numerical models of ice jams and intake blockage during the spring thaw at Bjoreio Dam, Norway, under different flow, gate conditions, and ice conditions. The simulations took between 11 hours and 10 days each to run. The faster run times were on a computer with 2 CPUs and 2 GB memory, while the slower runs were on a computer with 1 CPU and less than 1 GB memory (Basani 2006).

The studies described above include some general suggestions for working with general moving object models. FLOW-3D offers two or more solution methods for most physics models. In the case of the rigid body motion equations, the implicit method is more robust than the explicit method when the solid motion is coupled to the fluid motion (Basani 2006). Runtimes are lengthened by adding rigid body motion, and are best approached on computers that are newer and more powerful. Laptops are often prohibitively slow for these simulations (Basani 2006).

Modeling cavitation formation and damage for dams and tunnels with FLOW-3D

Cavitation is very difficult to model with scale physical models because the cavitation inception is subject to non-linear scale effects (Brennan 1995), and this makes it amenable to computational modeling. FLOW-3D has had an optional cavitation physics model since 1989, which models the formation, movement, and collapse of super-cavitation structures. Early applications of the model studied cavitation in pipe constrictions, in sloshing tanks, and around torpedoes and underwater missiles. Early validations compared model results to photographs of cavitation around stationary discs (Hirt and Harper 1989, Hirt 1990).

Because the cavitation model is very time-intensive and requires relatively fine mesh resolution (fine enough to resolve each cavity macrostructure), it is often impractical to use for spillway modeling. It is also unnecessary, as most cavitation studies need only determine the likelihood and locations of cavity formation, and do not require estimation of the size and collapse location of the cavities (Falvey 1990). For these reasons, dam and spillway simulations studying cavitation usually only require the pressure distribution output to identify cavitation potential. An example of this is the study of the Cabinet Gorge Dam by Groeneveld et al. (2007) described above. The full cavitation model appears unnecessary at this time unless the model domain is very small, as for an individual valve or other piece of equipment.

Modeling air entrainment with FLOW-3D

Air entrainment and its effects on fluid flow are important factors in some spillway cases, particularly where "bulking" of the flow (i.e. density changes in the bulk liquid/gas mixture) changes the flow direction. One of the most common cases involves a jetting discharge: if the jet density and momentum are altered by significant entrained air volume, the angle of the jet and subsequent downstream velocity profiles will also be affected. Not including the dynamics of air entrainment in these cases is a source of error (e.g. Cook and Richmond 2001).

Optional bulking and non-bulking turbulent air entrainment models were added to FLOW-3D in 2003. Early validations of the models included comparisons of numerical model results to experimental results for a vertical jet impinging on a pool, for hydraulic jumps in partially submerged conduits, for air entrainment in drop shafts, and for air entrainment on ogee spillways (Hirt 2003). The models were calibrated with an empirical rate coefficient, which was determined to best fit the data of all simulations when set to 0.5. Since 2003, 0.5 has been the default value for the rate coefficient, which can be adjusted by the user.

In 2005, the optional air entrainment models were extended to allow modeling of laminar air entrainment, reduce the occurrence of unphysical and excessive bulking, model air entrainment from trapped bubbles under pressure, and model bubble drag interactions and gas escape at free surfaces (Flow Science 2005b). These additions increased the power and range of application for the air entrainment model, but the extensions appear have lessened the accuracy of the default rate coefficient. Additional improvements in the turbulence models have also affected the calibration of the rate coefficient.

Cursory unpublished studies by the author suggest that the model results are sensitive to the rate coefficient, and 0.5 is sometimes, but not always, a good fit for hydraulic simulations. Similarly, when the air entrainment model is coupled with the optional multi-phase flow (drift flux) model to include bubble drag effects and gas escape, then the model results are very sensitive to the specified mean bubble radius. Test calibrations of the rate coefficient and mean bubble radius are suggested when air entrainment effects are included in numerical models.

NUMERICAL SIMULATION OF FLOW AND ABUTMENT SCOUR IN A COMPOUND CHANNEL

A series of 3-D numerical models of flow and scour in a compound channel with a rectangular abutment were developed using the commercial computational fluid dynamics (CFD) software package FLOW-3D. Results were compared to published data of Sturm (2004). This section includes a general discussion of compound channel hydraulics, followed by a description of the numerical models, and ends with comparison and discussion of physical and numerical model results. Reference is made to the above general discussion of CFD modeling of hydraulics, as the case studied below illustrates ongoing issues with turbulence modeling, sediment scour modeling, and simulation run times.

Introduction to compound channel hydraulics

Abutment scour in compound channels is a difficult and important problem for hydraulic engineers. As a result of several prominent bridge failures, the U.S. Federal Highway Administration (FHWA) has undertaken an effort to require all States to evaluate highway bridges for scour potential. Sturm (2004) gives a brief history of abutment scour failures, and discusses the difficulty in predicting abutment scour in detail. In short, the problem has two major aspects: the difficulty of modeling scour processes in general, and the difficulty in modeling flow in compound channels.

Most laboratory experiments resulting in empiric equations have assumed rectangular channels (Richardson 1993), although this is rarely the case in the field. A more typical real channel can be described as a compound channel, consisting of a main channel and a flood plain on one or two sides. The main channel carries the base flow and flow up to bank-full stage, while the flood plains carry excess flow. Flood plain flows are usually quite shallow compared to main channel flow.

In the field, any abutment may result in a number of different scour depths as a complex function of the approach flow distribution in the compound channel and the abutment length (Sturm 2004). Flow distribution in compound channels is affected by the interaction at an unfixed interface between main-channel flow and floodplain flow. The interface is characterized by vortexes and momentum exchange. Adding the calculated main-channel and floodplain flows separately over-predicts the actual discharge in compound channels. The lessened discharge capacity compared to the non-interacting component discharges is due to the head loss at the interface (e.g. Sturm 2004). The hydraulic situation becomes even more complex when an abutment blocks part or all of one or both flood plains.

Numerical models have been proposed to describe the interface dynamics, and have included 1-D, 2-D, and 3-D models (Sturm 2004). 1-D and 2-D models are generally problematic for these problems because they typically assume a vertically-parabolic velocity profile, and this assumption becomes invalid immediately downstream of bridge structures such as abutments where an elliptical profile with a recirculation zone forms. In cases where the width-to-depth ratio of the floodplain flow is large and vertical mixing is strong, the depth-averaged assumptions may be adequate (Sturm 2004).

Description of physical experiment

The physical experiment (Sturm 2004) took place in a flume 4.211 meter wide by 24.4 meter long with a fixed bed slope of 0.0022 m/m. The floodplain and main channel were constructed of poured lean-mix concrete with 3.3-mm gravel aggregate with geometric standard deviation $\sigma_g = [d_{84}/d_{16}]^{0.5} = 1.3$. A vertical abutment (0.80 meters long by 0.15 meters wide) was constructed using concrete blocks with centerline 9.75 meters from the flume entrance. The CFD model of the channel is shown in Figure 1.

Figure 1: Numerical model of compound channel and abutment

Flow rate and depth in the physical model were controlled by an upstream valve and a downstream gate. The flow rate *Q* was metered at 0.0850 m^3/second. Both the tail water depth just upstream of the gate y_{ftw} and the normal depth y_{f0} were set at 3.87 cm above the floodplain floor (19.27 cm above the main channel floor). The normal depth y_{f0} was determined to be a function of flow rate and was defined as the depth approached asymptotically by both flow-thickening and flow-thinning surface elevation profiles, and was measured from the flood plain floor without abutments in place.

After determining the normal depth, an abutment was added to the flood plain, and surface elevation profiles, velocity distributions, and flow partitioning between the main channel and flood plain were measured. Finally, a section of movable bed was added in the vicinity of the abutment. The bed was filled with the same uniform fine gravel used for the fixed bed concrete aggregate (d_{50} = 3.3 mm, σ_g = 1.3). In this final set of experiments, the equilibrium scour depth and time rate of scour formation were measured. For a complete description of the physical experiment and associated analysis, see Sturm (2004).

<u>Results of physical experiment</u>

Surface profiles were measured for the fixed bed case at 22 stations and presented as the average of 8 longitudinal measurements at each station. The resulting average surface profile was presented graphically in Sturm (2004), and was digitized using the free software *Engauge* (Figure 2 below) to allow quantitative comparison with CFD model results. The error introduced by digitization is estimated as ±0.6%. The sharp dip visible near station 9.8 corresponds to the location of the abutment which extends 0.80 meters into the floodplain.

Time rate of scour was measured for the movable bed case and reported non-dimensionally as a function of the ratio between *resultant* velocity at a point near the upstream abutment corner and the critical velocity for incipient sediment motion. Critical velocity was calculated using the Keulegan relationship as described in Sturm (2004). When included in the excess velocity ratio V_{ab}/V_c, the critical velocity is assumed to be calculated using the floodplain depth at the abutment face y_{ab}, based on the author's reading of Sturm (2004). Digitized time rate of scour data for two excess velocity ratios are reported in Figure 3.

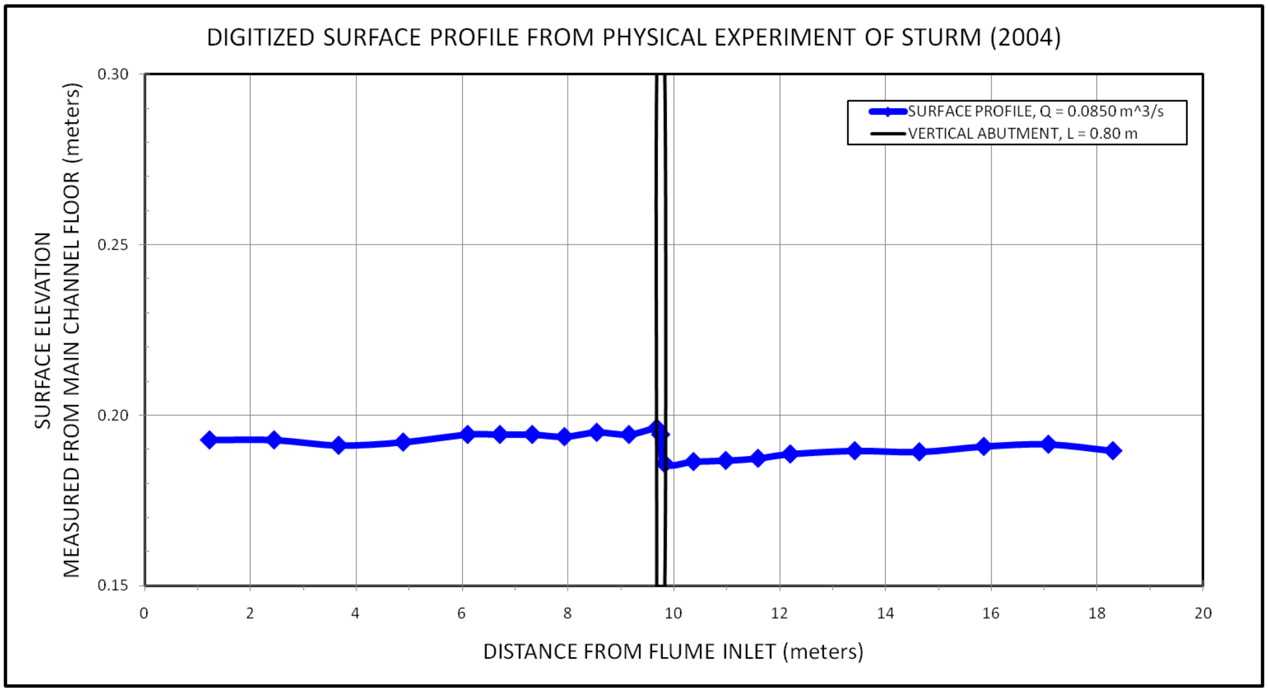

Figure 2: Digitized physical fixed bed average water surface profile

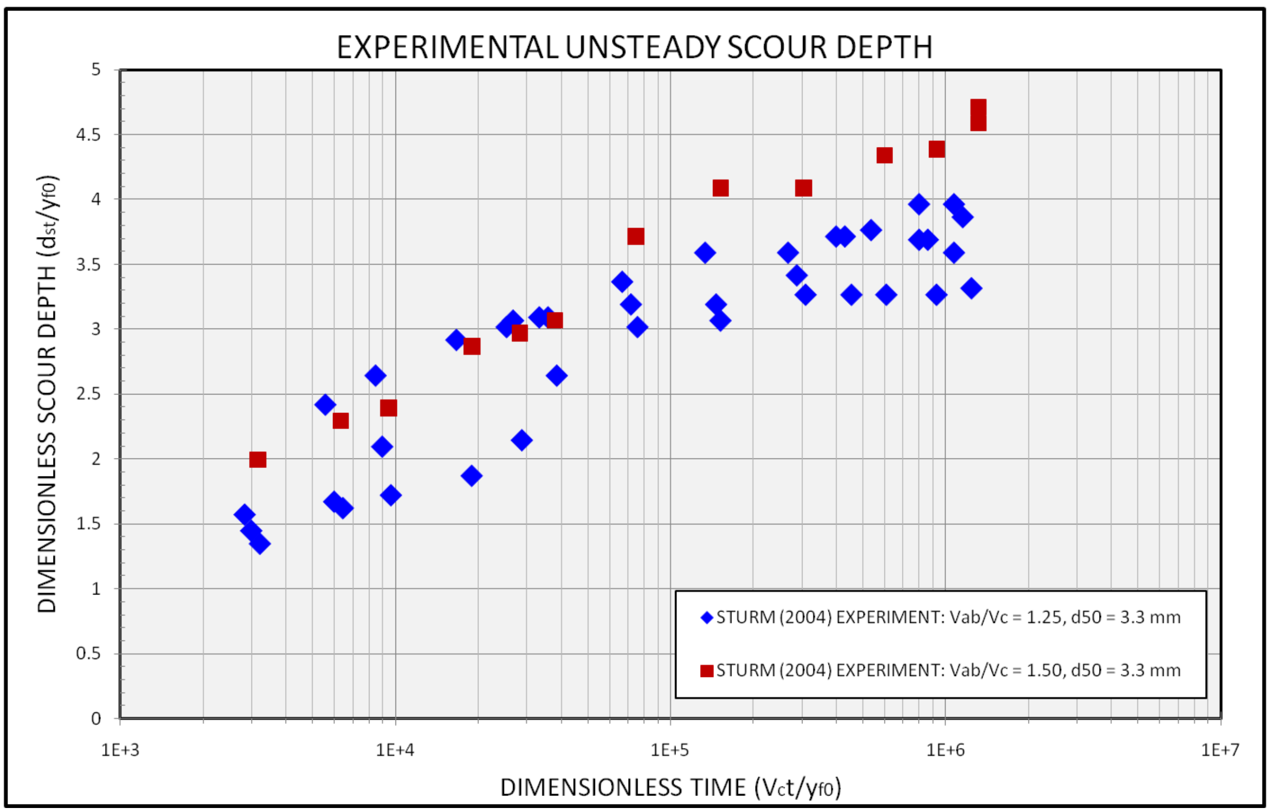

Figure 3: Digitized physical time rate of scour

Description of numerical experiments

A series of five numerical models were developed using FLOW-3D v10.0 to simulate the pre-scour (fixed bed) conditions reported in Figure 2 above. The model used the following options:

- TruVOF® method with free-surface tracking.
- Incompressible flow with 'limited' compressibility to model pressure shockwave propagation.
- Component gravity vectors in the negative z direction and positive x direction calculated to give the entire domain a bed slope S_b = 0.0022 m/m and gravitational acceleration magnitude g = 9.81 m/s^2.
- Renormalized Group (RNG) κ-ε turbulence model with dynamically computed maximum turbulent mixing length.
- Water assumed to be at 20° C (temperature not reported in physical study): density ρ_f = 998.23 kg/m^3, dynamic viscosity μ_f = 0.001 kg/m/s.
- Upstream boundary condition = volumetric flow rate type. Flow rate Q = 0.0850 m3/s, specified free surface elevation and location (station, in x) of the boundary as shown in Table 1.
- Downstream boundary condition and location (station in x) varied as shown in Table 1.
- Cell dimensions varied as shown in Table 1.
- Implicit Generalized Minimal Residual (GMRES) pressure solver.
- 2nd-order-monotonicity-preserving momentum advection equation.
- Finish time as shown in Table 1.
- Models run using FLOW-3D v10.0.k on 2 CPUs of a 6 GB, 3.2 GHz Intel Core i7 with 64-bit Windows 7 Ultimate.

		UPSTREAM (X-MIN) BOUNDARY			DOWNSTREAM (X-MAX) BOUNDARY				
CASE NUMBER	FINISH TIME (sec)	STA. X (m)	BOUNDARY TYPE	SPECIFIED DEPTH (cm)	STA. X (m)	BOUNDARY TYPE	SPECIFIED DEPTH (cm)	NUMBER OF MESH BLOCKS	CELL DIMENSIONS ∂x x ∂y x ∂z (cm)
1	1200	3.1	flow rate	19.19	21.4	outflow	n/a	1	5x5x0.5
2	1800	6	flow rate	19.19	14	pressure	18.96	1	2x2x1
3	1800	6	flow rate	19.28	14	pressure	19.27	1	2x2x1
4	1800	0	flow rate	19.28	14	pressure	19.27	2	4x4x2 & 2x2x1
5	400	0	flow rate	19.28	14	pressure	18.96	2	4x4x2 & 2x2x1

Table 1: Varied Parameters for Stage-1 (Hydraulic) CFD Simulations

Two additional models were developed to simulate the sediment scour processes. These models used the final flow conditions developed in case 5 as their starting conditions and upstream boundary conditions. The scour experiments used the following options:

- TruVOF method with free-surface tracking.
- Incompressible flow with 'limited' compressibility to model pressure shockwave propagation.
- Component gravity vectors in the negative z direction and positive x direction calculated to give the entire domain a bed slope S_b = 0.0022 m/m and gravitational acceleration magnitude g = 9.81 m/s^2.
- Renormalized Group (RNG) κ-ε turbulence model with dynamically computed maximum turbulent mixing length.
- Sediment scour model with sediment parameters varied as shown in Table 2.
- Water assumed to be at 20° C (temperature not reported in physical study): density ρ_f = 998.23

kg/m^3, dynamic viscosity μ_f = 0.001 kg/m/s.

- Upstream boundary condition = grid overlay, a spatially-varied, time-constant velocity profile interpolated from the Case 5 flow field at time t = 400 seconds and station x = 6 m downstream from flume entrance.
- Downstream boundary condition = constant hydrostatic pressure with surface elevation = 18.86 cm and location (station in x) = 14 m downstream from flume entrance.
- Cell dimensions = 2 cm x 2 cm x 1 cm (∂x x ∂y x ∂z).
- Implicit Generalized Minimal Residual (GMRES) pressure solver.
- 2nd-order-monotonicity-preserving momentum advection equation.
- Finish time = 3600 seconds.
- Models run using FLOW-3D/MP v4.2a on 16 CPUs of a multi-computer cluster with 64-bit Red Hat Enterprise Linux 5.0.

CASE NUMBER	SEDIMENT DIAMETER (mm)	SEDIMENT DENSITY (kg/m^3)	CRITICAL SHIELDS (dimless)	DRAG COEFFICIENT (dimless)	ENTRAINMENT COEFFICIENT (dimless)	BED LOAD COEFFICIENT (dimless)	ANGLE OF REPOSE (degrees)	PACKED POROSITY (%)
5r1a	3.3	2650	0.0470	1.5	0.018	5.7	35	30%
5r1b	3.3	2650	0.0236	1.5	0.018	5.7	35	30%
5r1c	3.3	2650	0.0470	1.5	0.018	0 (no bed load)	n/a	30%
5r1d	3.3	2650	0.0236	1.5	0.018	0 (no bed load)	n/a	30%

Table 2: Varied Parameters for Stage-2 (Sediment Scour) CFD Simulations

<u>Results of numerical experiments</u>

Average water surface profiles from the numerical experiments are shown below in Figure 4. Absolute and relative differences between physical and numerical results are given for each station reported in the physical study in Table 3. To compare the numeric models to each other, the mean error magnitudes between stations 6 and 14 are given in Table 4. The area of interest is between stations 6 and 14. This is the area affected by the abutment, and in the numerical models excludes boundary condition effects. For all models, the average error is less than ±2% and the maximum error for any average station depth is less than ±5%. The most accurate model is Case 5, illustrated in the area of interest in Figure 5.

The fixed bed flow field from Case 5 was used as the initial condition of the sediment scour models. The two scour models with critical Shields parameter τ_{*c} = 0.0470 determined directly from Shields diagram gave good results, while the two models with critical Shields parameter τ_{*c} = 0.0236 determined as one half of the Brownlie diagram (as recommended for engineering purposes in García 2008) significantly over-predicted erosion. Sturm (2004) used the Shields diagram as the basis for estimating the Shields parameter and critical velocity, and that approach is supported by the numerical results of the present study.

The scour simulations were not run to the expected equilibrium scour depth because doing so required more time than was available. Initial results for time-dependent (unsteady) scour depth d_{st} are closely within the scatter of physical experimental values when bed load transport is included (Case 5r1a), as shown in Figure F. When bed load transport is not included (Case 5r1c), scour depth is significantly under-predicted, as indicated for t = 3600 seconds in Figure 6.

The location of the maximum scour depth is unexpected, occurring at the edge of the movable bed area rather than at the tip of the abutment. Scour data reported in Sturm (2004) indicates that for several runs, similar results were seen. It is not clear if the results presented here are physically accurate, matching those cases where scour occurred due to transition effects between the movable and fixed bed regions, or whether the location of the scour pit is a numerical artifact related to the parameterization of the CFD model.

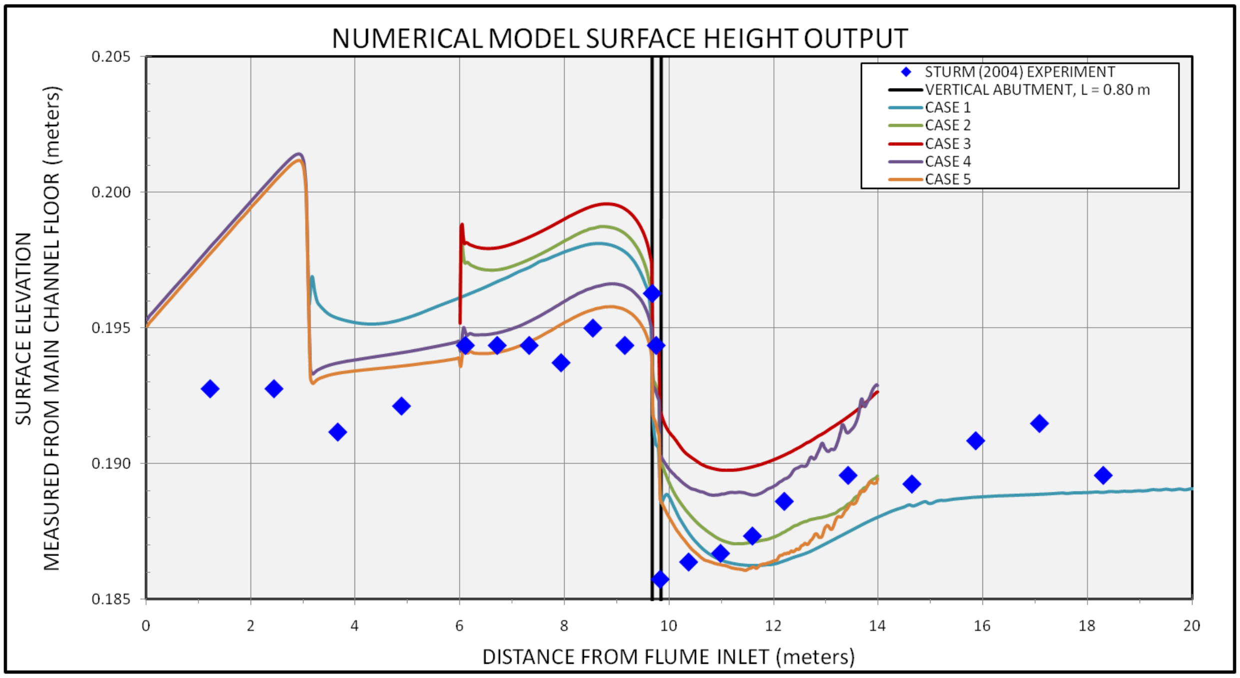

Figure 4: Comparison of all models and experiment: mean channel depth

FLUME STATION X (m)	AVERAGE DEPTH ABSOLUTE DIFFERENCE e_{abs} (cm) e_{abs} = (digitized physical - numerical)					AVERAGE DEPTH RELATIVE DIFFERENCE e_{rel} (%) $e_{rel} = e_{abs}$/(digitized physical) x 100				
	CASE 1	CASE 2	CASE 3	CASE 4	CASE 5	CASE 1	CASE 2	CASE 3	CASE 4	CASE 5
1.220	-	-	-	-0.52	-0.50	-	-	-	-2.71%	-2.58%
2.440	-	-	-	-0.78	-0.76	-	-	-	-4.07%	-3.95%
3.660	-0.42	-	-	-0.25	-0.22	-2.20%	-	-	-1.33%	-1.13%
4.880	-0.32	-	-	-0.20	-0.15	-1.66%	-	-	-1.02%	-0.77%
6.100	-0.18	-0.30	-0.38	-0.05	0.01	-0.94%	-1.56%	-1.93%	-0.27%	0.06%
6.710	-0.23	-0.28	-0.36	-0.05	0.03	-1.21%	-1.43%	-1.84%	-0.23%	0.13%
7.320	-0.29	-0.31	-0.40	-0.09	-0.01	-1.47%	-1.62%	-2.04%	-0.45%	-0.06%
7.930	-0.40	-0.44	-0.52	-0.22	-0.14	-2.08%	-2.27%	-2.69%	-1.13%	-0.73%
8.540	-0.31	-0.37	-0.45	-0.15	-0.07	-1.59%	-1.88%	-2.30%	-0.76%	-0.35%
9.150	-0.35	-0.41	-0.50	-0.22	-0.13	-1.78%	-2.12%	-2.59%	-1.12%	-0.68%
9.675	0.27	0.07	-0.04	0.19	0.29	1.36%	0.38%	-0.22%	0.95%	1.46%
9.750	0.36	0.15	0.00	0.17	0.29	1.88%	0.77%	0.01%	0.88%	1.50%
9.825	-0.38	-0.50	-0.68	-0.51	-0.35	-2.05%	-2.69%	-3.65%	-2.73%	-1.90%
10.370	-0.11	-0.18	-0.39	-0.28	-0.06	-0.59%	-0.96%	-2.11%	-1.48%	-0.33%
10.980	0.02	-0.05	-0.31	-0.22	0.04	0.13%	-0.27%	-1.65%	-1.16%	0.22%
11.590	0.11	0.02	-0.25	-0.15	0.12	0.58%	0.13%	-1.36%	-0.80%	0.62%
12.200	0.22	0.11	-0.17	-0.08	0.19	1.17%	0.61%	-0.90%	-0.43%	1.02%
13.420	0.21	0.11	-0.22	-0.16	0.12	1.11%	0.57%	-1.14%	-0.82%	0.63%
14.640	0.08	-	-	-	-	0.42%	-	-	-	-
15.860	0.21	-	-	-	-	1.09%	-	-	-	-
17.080	0.26	-	-	-	-	1.37%	-	-	-	-
18.300	0.06	-	-	-	-	0.33%	-	-	-	-

Table 3: Absolute and relative differences in experimental and predicted depths

MEAN AND MAXIMUM ERROR MAGNITUDE e_{mag} BETWEEN STATION 6.0 AND 14.0 m MEAN ERROR = sqrt[$\sum(e^2)/n$]; MAX ERROR = sqrt[e_{max}^2]					
	CASE 1	CASE 2	CASE 3	CASE 4	CASE 5
MEAN	0.27 1.40%	0.28 1.46%	0.38 1.98%	0.21 1.12%	0.17 0.89%
MAXIMUM	0.40 2.08%	0.50 2.69%	0.68 3.65%	0.51 2.73%	0.35 1.90%

Table 4: Overall magnitude of absolute and relative depth differences between stations 6 and 14

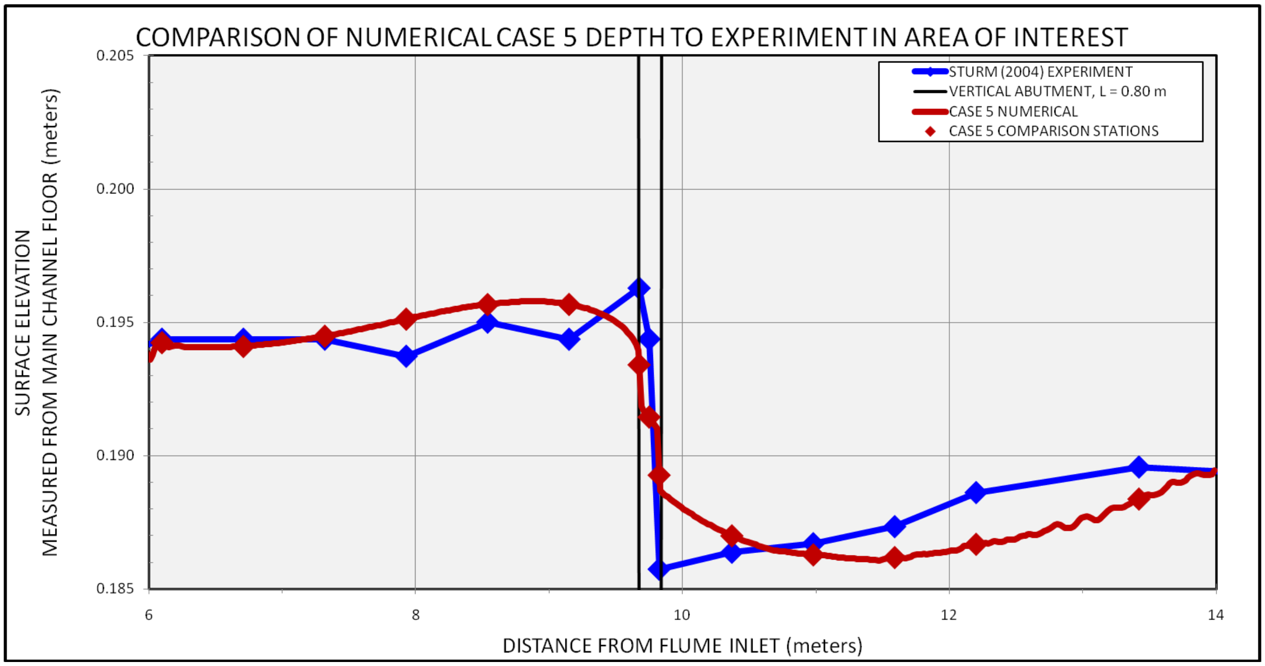

Figure 5: Comparison of Case 5 and experiment: mean channel depths between stations 6 and 14

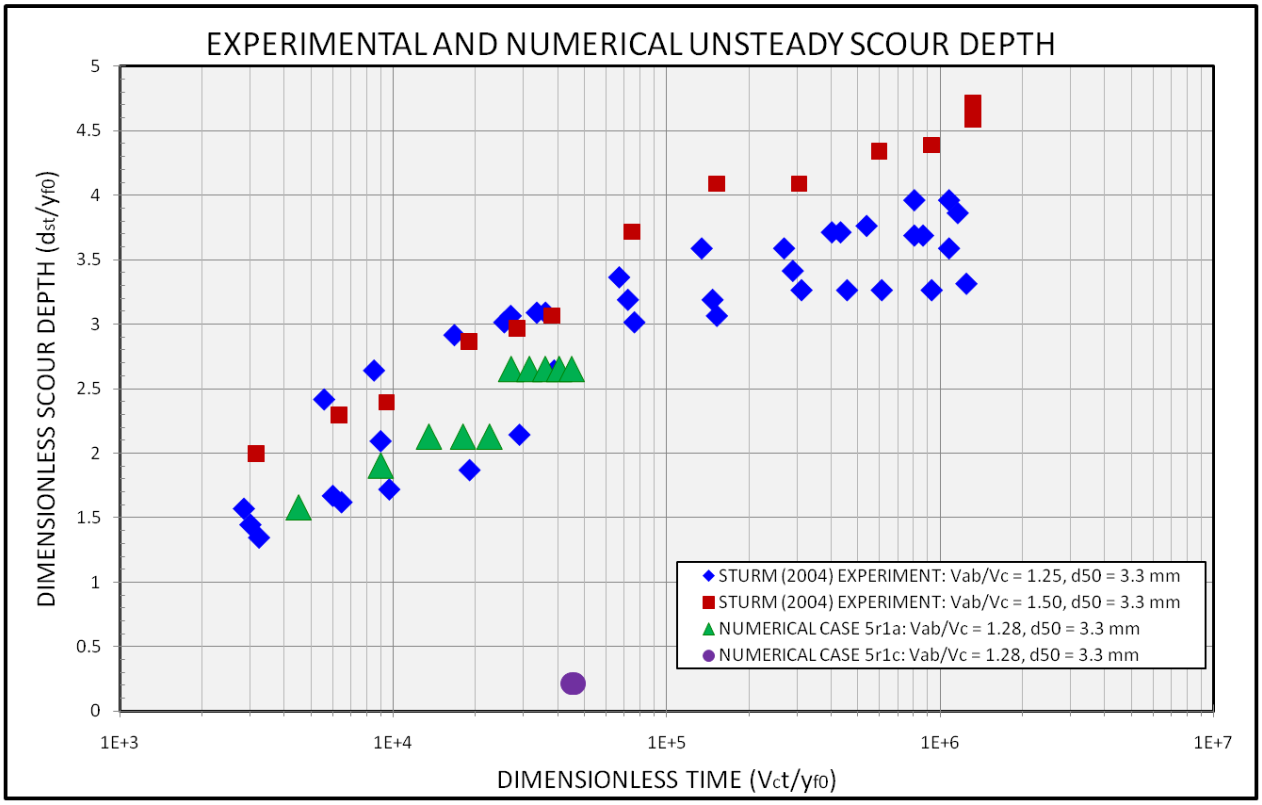

Figure 6: Comparison of Cases 5r1a, 5r1c, and experiment: time evolution of scour depth

Discussion of results

The numerical and experimental surface profile and time-rate-of-scour results are in good agreement. Although the scour pit occurs in an unexpected location in the numerical model, this may match at least some of the physical model results.

As indicated in Figure 5 and Table 3, the maximum disagreement in water surface elevation of 1.9% occurs at the point where the water surface 'hole' occurs immediately downstream of the abutment edge, with the numerical results indicating less change in depth than shown in the physical results. Inherent limitations of the κ-ε RNG turbulence model have been discussed above; all the κ-ε models are unable to completely resolve highly swirled flows and eddy structures, and that appears to be the main cause of the disagreement observed. It would be expected that results using a large eddy simulation (LES) turbulence model would more closely match the experimental data, and this is an avenue for further investigation.

As would be expected for fine gravel, bed load transport plays a significant role in scour formation for this case. Eliminating bed load transport causes scour to be under-predicted and including bed load transport (all other parameters being equal) appears to give scour evolution that agrees closely with physical experiment.

Currently, the FLOW-3D scour model is computationally intensive, and scour simulations are limited to single events. In the case of the current study, it was not feasible to simulate the entire physical experiment (nearly 30 hours). The drawback of only modeling the first hour of scour is that the equilibrium scour depth remains uncalculated and cannot be compared with physical experiment. One possible solution to this problem is to use the entrainment rate coefficient as a time-scaling factor. The potential applicability of this approach has been demonstrated (e.g. Lin et al. 2010), but the limitations of time-scaling have not been thoroughly evaluated.

Summary of CFD Scour Model

A series of nine CFD models were developed to compare numerical results to physical results for fixed bed and movable bed compound channel flow with a vertical-wall abutment. The numerical models showed good agreement with the physical results of Sturm (2004), although equilibrium scour depth could not be achieved without time scaling. Analysis of the results indicate that the κ-ε RNG turbulence model under-predicts the depth of draw immediately downstream of the abutment, although the disagreement observed was less than 1.9%. The applicability of time-scaling of scour phenomena needs to be evaluated for future studies since it appears that 1:1 numerical-to-physical scour rates are time-prohibitive for many industrial applications.

SUMMARY

It is the author's hope that the literature review and general discussion presented above will be helpful to current and future CFD modelers working on dam and river projects. The main applications of FLOW-3D are spillway design and analysis, estimation of nappe impingement location and resulting velocity fields, estimation of pressure distribution and tractive force on hydraulic structures, design and analysis of tailraces and stilling basins, and river reach and intake design and analysis. For these types of problems, FLOW-3D has a strong history of industry use, with very good published comparisons between physical and numerical models. Modelers should still be aware of the limitations of the most popular turbulence closures and should watch out for numerical diffusion due to elongated cell aspects and inter-block boundary cell ratios. Up-and-coming model applications of FLOW-3D to river and dam simulations include sediment scour modeling, moving objects, cavitation, and air entrainment. For these models, users should be aware of longer run times and ongoing development of the software. Finally, a new simulation of sediment scour was presented that agrees favorably with the physical experiment for both surface profile and scour depth comparisons. The model illustrates the limitations of the RNG turbulence model, and also shows how the limitations are not severe enough to demand a more computationally expensive model.

REFERENCES

Abad, J.D. and García, M.H. (2005). "Hydrodynamics in Kinoshita-Generated Meandering Bends: Importance for River-Planform Evolution," *River, Coastal and Estuarine Morphodynamics: Proc. of the 4th IAHR Symposium on River, Coastal and Estuarine Morphodynamics (RCEM 2005, Urbana, IL, USA, Oct 4-7, 2005), Vol 1*, Parker, G., ed., Taylor and Francis Group, London, UK.

Abad, J.D., Rhoads, B.L., Güneralp, İ., and García, M.H. (2008). "Flow Structure at Different Stages in a Meander-Bend with Bendway Weirs." *Journal of Hydraulic Engineering* 134(8): 1052-1063.

Amiraslani, S., Fahimi, J., and Mehdinezhad, H. (2010). "The Numerical Investigation of Free Falling Jet's Effect on the Scour of Plunge Pool." *Proc. XVIII International Conference on Computational Methods in Water Resources, Barcelona, Spain, Jun 21-24, 2010*, J. Carrera, ed., CIMNE, Barcelona.

Amorim, J.C.C., Rodrigues, C.R., and Marques, M.G. (2004). "A Numerical and Experimental Study of Hydraulic Jump Stilling Basin," *Advances in Hydro-Science and-Engineering, Volume VI: Proceedings of Abstracts and Papers of the 6th International Conference on Hydro-Science and-Engineering, Brisbane, Australia, May 31-Jun 3, 2004*, Wang, S.S., ed., University of Mississippi, Oxford, MS.

Basani, R. (2006). “Numerical Simulation of a Hydraulic Power Intake – Ice Modeling.” *Proc. 2006 FLOW-3D World User’s Conference, Denver, CO, Sep 21-26, 2006*, Flow Science Inc., Santa Fe, NM.

Bombardelli, F.A., Meireles, I., and Matos, J. (2010). “Laboratory Measurements and Multi-Block Numerical Simulations of the Mean Flow and Turbulence in the Non-Aerated Skimming Flow Region of Steep Stepped Spillways.” *Environmental Fluid Mechanics*, 10(4): 263-288.

Bouton, M. (1991). “Influence of trapped gas on water hammer effects in tube.” *ESA, Aerothermodynamics for Space Vehicles*, pp 509-513, Aerospatiale, Paris, 1991.

Brennan, C.E. (1995). *Cavitation and Bubble Dynamics*. Oxford University Press: New York, NY.

Briand, M., Tremblay, C., Bossé, Y., Gacek, J., Alfaro, C., and Blanchet, R. (2010). “Ashlu Creek Hydroelectric Project: Design and Optimization of Hydraulic Structures Under Construction Using 2D and 3D Numerical Modeling.” *Canadian Dam Association 2010 Annual Conference*, Niagara Falls, ON, Canada, Oct 2-7, 2010.

Brown, D.S., MacDonell, D., Sydor, K., and Barnes, N. (2009). “An Integrated Computational Fluid Dynamics and Fish Habitat Suitability Model for the Pointe Du Bois Generating Station.” *Canadian Dam Association 2009 Annual Conference*, Whistler, BC, Canada, Oct 3-8, 2009.

Chanel, P.G. and Doering, J.C. (2007). “An Evaluation of Computational Fluid Dynamics for Spillway Modeling.” *16th Australasian Fluid Mechanics Conference*, Gold Coast, Australia, Dec 3-7, 2007.

Chanel, P.G. and Doering, J.C. (2008). “Assessment of Spillway Modeling Using Computational Fluid Dynamics.” *Canadian Journal of Civil Engineering* 35: 1481-1485.

Cho, S. (2010). “Foundation Design of the Incheon Bridge.” *Geotechnical Engineering Journal of the SEAGS & AGSSEA* 41(4).

Choi, J., Ko, K.O., and Yoon, S.B. (2009). “3D Numerical Simulation for Equivalent Resistance Coefficient for Flooded Built-Up Areas.” *Proc. of the 5th International Conference on Asian and Pacific Coasts*, Singapore, Oct 13-16, 2009.

Chow, V.T. (1959). *Open Channel Hydraulics*, McGraw-Hill, New York, NY.

Cook, C.B. and Richmond, M.C. (2001). “Simulation of Tailrace Hydrodynamics Using Computational Fluid Dynamics Models.” *US Army Corp of Engineers Portland District Report PNNL-13467*, U.S. Dept of Commerce, National Technical Information Service, Springfield, VA.

Cook, C.B., and Richmond, M.C. (2004). “Monitoring and Simulating 3-D Density Currents and the Confluence of the Snake and Clearwater Rivers.” *Critical Transitions In Water And Environmental Resources Management: Proceedings Of The 2004 World Water and Environmental Resources Congress, Jun 27-Jul 1, 2004, Salt Lake City, UT*, G. Sehlke et al., eds., American Society of Civil Engineers, Reston, VA.

Cook, C.B., Richmond, M.C., Serkowski, J.A., and Ebner, L.L. (2002). "Free-Surface Computational Fluid Dynamics Modeling of a Spillway and a Tailrace: Case Study of The Dalles Project." *Proc. of Hydrovision 2002 Conference*, Portland, OR, July 29-Aug 2, 2002.

Dargahi, B. (2010). "Flow Characteristics of Bottom Outlets with Moving Gates." *Journal of Hydraulic Research* 48(4): 476-482.

Dworak, F.J. (2005). "Characterizing Turbulence Structure along Woody Vegetated Banks in Incised Channels: Implications for Stream Restoration," thesis: The University of Tennessee, Knoxville, TN, Dec 2005.

Echols, J.F., Pratt, M.A., and Williams, K.A. (1998). "Using CFD to Model Flow in Large Circulating Water Systems." *Proceedings of the 1998 POWER-GEN International Conference*, Orlando, FL, Dec. 9-11, 1998.

Falvey, H.T. (1990). *Cavitation in Chutes and Spillways*. *Water Resources Technical Publication, Engineering Monograph #42*. U.S. Dept of the Interior, Bureau of Reclamation, Denver Office, Denver, CO.

Flow Science, Inc. (2005a). "Using FLOW-3D to Model Valves and Water Hammer Effect," *FLOW-3D News*, Summer 2005, Flow Science, Inc., Santa Fe, NM.

Flow Science, Inc. (2005b). "Extensions to the Air Entrainment Model (or Passing Gas Discretely)." *FLOW-3D News*, Summer 2005, Flow Science, Inc., Santa Fe, NM.

Fuamba, M., Brousseau, G., Mainville, E., Colameo, S., and Sparks, D. (2006). "Role and Behavior of Surge Chamber in Hydropower: Case of the Robert Bourassa Hydroelectric Power Plant in Quebec, Canada." *Dams and Reservoirs, Societies and Environment in the 21st Century*, L. Berga et al., eds., Taylor & Francis Group, London, UK.

García, M.H. (2008). "Sediment Transport and Morphodynamics." *Sedimentation Engineering*, M.H. García, ed., American Society of Civil Engineers, Reston, VA.

Gendzelevich, W., Baryla, A., Groenveld, J., and McNeil, D. (2009). "Red River Floodway Expansion Project-Design and Construction of the Outlet Structure," *Canadian Dam Association 2009 Annual Conference*, Whistler, BC, Canada, Oct 3-8, 2009.

Gessler, D. (2005). "CFD Modeling of Spillway Performance," *Proceedings of the 2005 World Water and Environmental Resources Congress, May 15-19, 2005, Anchorage, Alaska*, Walton, R., ed., American Society of Civil Engineers, Reston, VA.

Gessler, D. and Rasmussen, B. (2005). "Before the Flood." *Desktop Engineering*, Oct 2005.

Griffith, A.R., Rutherford, J.H., Alavi, A., Moore, D.D., and Groeneveld, J. (2007). "Stability Review of the Wanapum Spillway Using CFD Analysis." *Canadian Dam Association Bulletin*, Fall 2007.

Groeneveld, J. (1999). "Computer Simulation Leads to Faster, Cheaper Options," *Water Engineering & Management*, Jun 1999: 14-17.

Groeneveld, J., Sweeney, C., Mannheim, C., Simonsen, C., Fry S., and Moen, K. (2007). “Comparison of Intake Pressures in Physical and Numerical Models of the Cabinet Gorge Dam Tunnel.” *Waterpower XV*, Chattanooga, TN, USA, Jul 23-26, 2007.

Groeneveld, J.L., Sydor, K.M., Fuchs, D.M., Teklemariam, E., and Korbaylo, B.W. (2001). “Optimization of Hydraulic Design Using Computational Fluid Dynamics.” *Proc. of Waterpower XII*, Salt Lake City, UT, Jul 9-11, 2001.

Heiner, B. and Barfuss, S.L. (2011). “Parshall Flume and Discharge Corrections Wall Staff Gauge and Centerline Measurements.” *Journal of Irrigation and Drainage Engineering,* online ahead of print, Feb 1, 2011. American Society of Civil Engineers, Reston, VA.

Hirsch, C. (2007). *Numerical Computation of Internal and External Flows*. 2nd edition, Elsevier, Burlingame, MA.

Hirt, C.W. (1990). "Computational Modeling of Cavitation." *Proc. 2nd International Symposium on Performance Enhancement for Marine Applications, Newport, RI, Oct 14-16, 1990*.

Hirt, C.W. (1994). "Weir Discharges and Counter Currents." *Proc. 1st International Conference on Hydroinformatics, Delft, Netherlands, Sep 19-23*.

Hirt, C.W. (2003). “Modeling Turbulent Entrainment of Air at a Free Surface,” *Technical Note FSI-03-TN61*, Flow Science Inc., Santa Fe, NM.

Hirt, C.W. and Harper, R.P. (1989) "Modeling Cavitation Phenomena." *Technical Note FSI-89-TN21*, Flow Science Inc., Santa Fe, NM.

Hirt, C.W. and Nichols, B.D. (1981). "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries," *Journal of Computational Physics* 39: 201.

Hirt, C.W., and Williams, K.A. (1994). "FLOW-3D Predictions for Free Discharge and Submerged Parshall Flumes." *Technical Note FSI-94-TN40*, Flow Science Inc., Santa Fe, NM.

Ho, D., Boyes, K., Donohoo, S., and Cooper, B. (2003b). “Numerical Flow Analysis for Spillways.” *Proc. 43rd ANCOLD Conference*, Hobart, Tasmania, Oct 24-29, 2003: 227-237.

Ho, D.K.H., Boyes, K. Donohoo, S.M. (2001). “Investigation of Spillway Behaviour Under Increased Maximum Flood by Computational Fluid Dynamics Technique.” *Proc. of the 14th Australasian Fluid Mechanics Conference*, Adelaide, Australia, Dec 10-14, 2001: 577-580.

Ho, D.K.H., Cooper, B.W., Riddette, K.M., and Donohoo, S.M. (2006). “Application of Numerical Modelling to Spillways in Australia.” *Dams and Reservoirs, Societies and Environment in the 21st Century: Proc. of the International Symposium on Dams in the Societies of the 21st Century, 22nd International Congress on Large Dams (ICOLD), Barcelona, Spain, June 18, 2006*, L. Berga et al., eds., Taylor & Francis Group, London, UK.

Ho, D.K.H., Donohoo, S.M., Boyes, Riddette, K.M., and Lock, C.C. (2003a). “Numerical Analysis and the Real World: It Looks Pretty But Is It Right?” *Proc. of the NAFEMS World Congress*, Orlando, FL, May 27-31, 2003.

Ho, J., Coonrod, J., Hanna, L. J., and Mefford, B. W. (2011). “Hydrodynamic Modelling Study of a Fish Exclusion System for a River Diversion.” *River Research and Applications* 27(2): 184-192.

Ho, J., Hanna, L., Mefford, B., and Coonrod, J. (2006). “Numerical Modeling Study for Fish Screen at River Intake Channel.” *Conference Proc. of World Water and Environmental Resources Congress 2006*, ASCE, Omaha, NE, USA, May 21-25, 2006.

Ho, J., Yeo, H.K., Coonrod, J., Ahn, W.-S. (2007). “Numerical Modeling Study for Flow Pattern Changes Induced by Single Groyne.” *2007 IAHR Congress Proc.*, Venice, Italy, Jul 1-6, 2007.

Ishino, K., Otani, H., Okada, R., and Nakagawa, Y. (1993). "The Flow Structure Around a Cylindrical Pier for the Flow of Transcritical Reynolds Number." *Proc. of the 25th Congress of the International Association for Hydraulic Research, Tokyo, Japan, Vol 5*, pp. 417-424, IAHR, Madrid, Spain.

Jacobsen, J. and Olsen, N. R. B. (2010). “Three-Dimensional Numerical Modeling of the Capacity for a Complex Spillway.” *Proc. of the ICE - Water Management* 163(6): 283-288.

Johnson, M.C. and Savage, B. (2006). “Physical and Numerical Comparison of Flow over Ogee Spillway in the Presence of Tailwater.” *Journal of Hydraulic Engineering* 132(12).

Kent, E.J., and Richardson, J.E. (1999). "Three-Dimensional Hydraulic Analysis for Calculation of Scour at Bridge Piers with Fender Systems." *Stream Stability and Scour at Highway Bridges*, E.V. Richardson and P.F. Lagasse, eds., American Society of Civil Engineers, Reston, VA.

Kim, D-G. (2007). “Numerical Analysis of Free Flow Past a Sluice Gate.” *KSCE Journal of Civil Engineering* 11(2): 127-132.

Kim, S., Lee, H., and An, S. (2010). “Improvement of hydraulic stability for spillway using CFD model.” *International Journal of the Physical Sciences* 5(6): 774-780.

Kitamura, Y., Kato, T., Kitamura, P. (2001). “Mathematical Modeling for Fish Adaptive Behavior in a Current.” *Proc. of the Third International Symposium on Environmental Hydraulics*, Tempe, AZ, Dec 5-8, 2001.

Kocamana, S., Seckinb, G., and Erduran, K.S. (2010). “3D model for prediction of flow profiles around bridges.” Journal of Hydraulic Research 48(4): 521-525.

Krishnappan, B.G., and Lau, Y.L. (1986). “Turbulence Modeling of Flood Plain Flows.” *Journal of Hydraulic Engineering*, 112(4): 251-266, American Society of Civil Engineers, Reston, VA.

Lavedrine, I., and Woolf, D. (2001). “Application of CFD Modelling to Hydraulic Structures.” *Proc. of Computing and Control in the Water Industry (CCWI) 2001*, De Montfort University, Leicaster, UK, Sep 3-5, 2001.

Lin, Y.-C., Capart, H., and Young, D.-L. (2010). “Local Bridge Scour in Alluvial River.” Flow Science Inc., Santa Fe, NM.

Lyn, D.A. (2008). “Turbulence Models for Sediment Transport Engineering.” *Sedimentation Engineering*, M.H. García, ed., American Society of Civil Engineers, Reston, VA.

McPhail, G., Lacelle, J., Smith, B., and MacMillan, D. (2010). “Upgrading of Boundary Dam Spillway.” *Canadian Dam Association 2010 Annual Conference*, Niagara Falls, ON, Canada, Oct 2-7, 2010.

Mefford, B. and Higgs, J. (2006). “Link River Falls Passage Investigation - Flow Velocity Simulation.” *Water Resources Research Laboratory Technical Paper - 954*, Feb 2006.

Meireles, I., Bombardelli, F.A., and Matos, J. (2010). “Experimental and Numerical Investigation of the Non-Aerated Skimming Flow on Stepped Spillways Over Embankment Dams.” *The 2010 IAHR European Congress*, Edinburgh, UK, May 4-6, 2010.

Miyamoto, Y., and Ishino, K. (1995). "Three Dimensional Flow Analysis in Open Channel," *Hydra 2000: Proceedings of the XXVIth Congress of the International Association for Hydraulic Research, London, 11-15 September 1995, Vol. 1*, Thomas Telford Ltd., London, UK.

Möller, G., Boes, R., Theiner, D., Fankhauser, A., De Cesare, G., and Schleiss, A. (2011). “Hybrid Modeling of Sediment Management During Drawdown of Räterichsboden Reservoir.” *Dams and Reservoirs under Changing Challenges*, A. Schleiss & R. Boes, eds., Taylor & Francis Group, London.

Mueller, D.S. and Wagner, C.R. (2007). “Correcting Acoustic Doppler Current Profiler Discharge Measurements Biased by Sediment Transport.” *Journal of Hydraulic Engineering* 133(12): 1329-1336.

Naot, D., Nezu, I., and Nakagawa, H. (1993). “Hydrodynamic Behavior of Compound Rectangular Open Channels.” *Journal of Hydraulic Engineering*, 119(3): 390-408, American Society of Civil Engineers, Reston, VA.

Nichols, B.D., and Hirt, C.W. (1975). "Methods for Calculating Multi-Dimensional, Transient Free Surface Flows Past Bodies." *Proc. First International Conference on Numerical Ship Hydrodynamics, Gaithersburg, ML, Oct. 20-23, 1975*.

Oertel, M., Heinz, G. and Schlenkhoff, A. (2010), “Physical and Numerical Modelling of Rough Ramps and Slides.” *The 2010 IAHR European Congress*, Edinburgh, UK, May 4-6, 2010.

Oertel, M. and Schlenkhoff, A. (2008). “Flood Wave Propagation and Flooding of Underground Facilities.” *International Conference on Fluvial Hydraulics (River Flow 2008)*, Izmir-Cesme, Turkey, Sep 3-5, 2008.

Ozmen-Cagatay, H. and Kocaman, S. (2010). “Dam-break flows during initial stage using SWE and RANS approaches.” *Journal of Hydraulic Research* 48(5): 603-611.

Paxson, G. and Savage, B. (2006). “Labyrinth Spillways: Comparison of Two Popular U.S.A. Design Methods and Consideration of Non-standard Approach Conditions and Geometries.” *International*

Junior Researcher and Engineer Workshop on Hydraulic Structures, Report CH61/06, Div. of Civil Eng., The University of Queensland, Brisbane, Australia.

Prinos, P. (1990). "Turbulence Modeling of Main Channel-Floodplain Flows With an Algebraic Stress Model." *Proc. of the International Conference on River Flood Hydraulics, Wallingford, U.K., Sep. 17-20, 1990*, W.R. White, ed., John Wiley & Sons, pp. 173-186.

Qiu, J. (2008). "Gravel Transport Estimation and Flow Simulation Over Low-Water Stream Crossings." thesis: Lamar University – Beaumont, AAT 3415945.

Raiford, J.P. and Khan, A.A. (2006). "Numerical Modeling of Internal Flow Structure in Submerged Hydraulic Jumps." *Conference Proc. of World Water and Environmental Resources Congress 2006*, ASCE, Omaha, NE, USA, May 21-25, 2006.

Richardson, J.E. (1997) "Control of Hydraulic Jump by Abrupt Drop." *Water for a Changing Global Community: Proc. of the 27th Congress of the International Association of Hydraulic Research, San Francisco, CA, Aug 10-15, 1997*, F.M. Holly and A. Alsaffar, eds., American Society of Civil Engineers, New York, NY.

Richardson, J.E. (2007). "CFD Saves the Alewife: Computer simulation helps the Alewife return to its Mt. Desert Island spawning grounds." *Desktop Engineering*, Jul 2007; *Hatchery International*, Jul/Aug 2007.

Richardson, J. and Dixon, D. (2004). "Modeling the Hydraulics Zone of Influence of Connecticut Yankee Nuclear Plant's Cooling Water Intake Structure." *The Connecticut River Ecological Study (1965-1973) Revisited: Ecology of the Lower Connecticut River 1973-2003*, P.M. Jacobson et al., eds., American Fisheries Society, Bethesda, MD.

Richardson, J.E., Panchang, V.G., and Kent, E. (1995). "Three-Dimensional Numerical Simulation of Flow Around Bridge Sub-Structures." *Water Resources Engineering: Proc. of the First International Conference, San Antonio, TX, Aug 14-18 1995*, W.H. Espey and P.G. Combs, eds., American Society of Civil Engineers, New York, NY.

Richardson, J.R., and Richardson, E.V. (1993). "Discussion of *Local Scour at Bridge Abutments* by B.W. Melville." *Journal of Hydraulic Engineering*, 119(9): 1069-1071, American Society of Civil Engineers, Reston, VA.

Rodi, W. (2010). "Large Eddy Simulation of River Flows." *Proc. Fifth International Conference on Fluvial Hydraulics (River Flow 2010)*, Braunschweig, Germany, Oct 8-10, 2010.

Rodriguez, J.F., Bombardelli, F.A., García, M.H., Frothingham, K., Rhoads, B.L., and Abad, J.D. (2004). "High-Resolution Numerical Simulation of Flow Through a Highly Sinuous River Reach." *Water Resources Management* 18:177-199.

Rodríguez, J.F., García, M.H., Bombardelli, F.A., Guzmán, J.M., Rhoads, B.L., and Herricks, E. (2000). "Naturalization of urban streams using in-channel structures." *Building Partnerships: Proc. of the 2000 Joint Conference on Water Resources Engineering and Water Resources Planning & Management, Minneapolis, MN, Jul 30-Aug 2, 2000*, American Society of Civil Engineers, Reston, VA.

Rüther, N., Jacobsen, J., Reidar N., Olsen, B., and Vatne, G. (2010). "Prediction of the Three-Dimensional Flow Field and Bed Shear Stresses in a Regulated River in Mid-Norway." *Hydrology Research* 41(2): 145-152.

Savage, B., Frizell, K., and Crowder, J. (2004). "Brains versus Brawn: The Changing World of Hydraulic Model Studies." *PAP-933*, U.S. Bureau of Reclamation, Hydraulic Investigations and Laboratory Services Group, Denver CO.

Savage, B.M. and Johnson, M.C. (2001). "Flow Over Ogee Spillway: Physical and Numerical Model Case Study." *Journal of Hydraulics Engineering* 127(8): 640-649, American Society of Civil Engineers, Reston, VA.

Savage, B.M., Johnson, M.C., and Towler, B. (2009). "Hydrodynamic Forces on a Spillway: Can We Calculate Them?" *Dam Safety 2009*, Hollywood, FL, USA, Sep 27-Oct 1, 2009.

Sahoo, G.B., Bombardelli, F., Behrens, D. and Largier, J.L. (2010). "Estimation of Stratification and Mixing of a Closed River System Using *FLOW-3D*." *American Geophysical Union, Fall Meeting 2010*, abstract #H31G-1091.

Sicilian, J.M. (1987). "A FLOW-3D Model for Flow in a Water Turbine Passage." *Report FSI-87-36-1*, Flow Science Inc., Santa Fe, NM.

Sick, M., Keck, H., Vullioud, G. and Parkinson, E. (2000). "New Challenges in Pelton Research." *Making Hydro More Competitive: Proc. of the Hydro 2000 Conference, Bern, Switzerland, Oct 2000*, Aqua-Media International, Surrey, UK.

Smith, H.D. (2007). "Flow and Sediment Dynamics Around Three-Dimensional Structures in Coastal Environments." thesis: The Ohio State University.

Spasojevic, M., and Holly, F.M. (2008). "Two- and Three-Dimensional Numerical Simulations of Mobile-Bed Hydrodynamics and Sedimentation." *Sedimentation Engineering*, M.H. García, ed., American Society of Civil Engineers, Reston, VA.

Sturm, T.W. (2004). "Enhanced Abutment Scour Studies for Compound Channels." *Publication No. FHWA-RD-99-156*, U.S. Dept of Transportation, Federal Highway Administration, McClean, VA.

Sweeney, C., Moen, K., and Kirschbaum, D. (2009). "Hydraulic Design of Total Dissolved Gas Mitigation Measures for Boundary Dam." *Waterpower XVI*, Spokane, WA, USA, Jul 27-30, 2009.

Sydor, K.M. and Waterman, P.J. (2010). "Engineering and Design: The Value of CFD Modeling in Designing a Hydro Plant." *Hydro Review* 29(6).

Teklemariam, E., and Groeneveld,, J.L. (2000). "Solving Problems in Design and Dam Safety with Computational Fluid Dynamics." *Hydro Review* 19(3): 48-52.

Teklemariam, E., Korbaylo, B.W., Groeneveld, J.L. and Fuchs, D.M. (2002). "Computational Fluid Dynamics: Diverse Applications In Hydropower Project's Design and Analysis." *Water Management* *in a Changing Climate: Proceedings of the 55th Canadian Water Resources Association (CWRA) 2002 Annual Conference*, Winnipeg, Manitoba, Jun 11-14, 2002.

Teklemariam, E., Shumilak, B., Murray, D., and Holder, G.K. (2008). “Combining Computational and Physical Modeling to Design the Keeyask Station.” *Hydro Review* 27(4).

Thomas, W.A., and Chang, H. (2008). “Computational Modeling of Sedimentation Processes.” *Sedimentation Engineering*, M.H. García, ed., American Society of Civil Engineers, Reston, VA.

United States Army Corp of Engineers (USACE, 1995). *Hydraulic Design of Spillways*, *Technical Engineering and Design Guides as Adapted from the US Army Corps of Engineers, No. 12*, American Society of Civil Engineers, Reston, VA.

Vasquez, J. (2010). “Assessing Sediment Movement by CFD Particle Tracking.” *The 2nd Joint Federal Interagency Conference on Sedimentation and Hydrological Modeling*, Las Vegas, NV, Jun 27-Jul 1, 2010.

Vasquez, J.A. and Roncal, J.J. (2009). “Testing River2D and *FLOW-3D* for Sudden Dam-Break Flow Simulations.” *Canadian Dam Association 2009 Annual Conference*, Whistler, BC, Canada, Oct 3-8, 2009.

Vasquez, J.A. and Walsh, B.W. (2009). “CFD Simulation of Local Scour in Complex Piers Under Tidal Flow.” *33rd IAHR Congress: Water Engineering for a Sustainable Environment*, ISBN: 978-94-90365-01-1.

Wang, G., Song, C.R., Kim, J., and Alexander, H.C. (2009). “Numerical Study of Erosion-Proof of Loose Sand in an Overtopped Plunging Scour Process—*FLOW-3D*.” *The 2009 Joint ASCE-ASME-SES Conference on Mechanics and Materials*, Blacksburg, VA, Jun 24-27, 2009.

Wang, S.M. and Sydor, K. (2008). “Power Intake Velocity Modeling using *FLOW-3D* at Kelsey Generating Station.” *CDA 2008 Annual Conference*, Winnipeg, MB, Canada, Sep 27-Oct 2, 2008.

Wang, S.M. and Sydor, K. (2010). “Power Intake Velocity Modeling Using *FLOW-3D* at Kelsey Generating Station.” *Canadian Dam Association Bulletin* 21(2): 16-21.

Williams, K.A., Diaz-Tous, I.A., and Ulovg, P. (1999). "Reduction in Pumping Power Requirements of the Circulation Water (CW) System at TU Electric's Martin Lake Plant Using Computation Fluid Dynamics (CFD)." *ASME Mechanical Engineering*, Jan. 1999.

Woolgar, R. and Eddy, W. (2006). “Using Computational Fluid Dynamics to Address Fish Passage Concerns at the Grand Falls-Windsor Hydroelectric Development.” *Canadian Dam Association 2006 Annual Conference*, Quebec City, Canada, Sep 30 – Oct 6, 2006.

Yoo, S., Hong, K., and Hwang, M. (2002). “A 3-Dimensional Numerical Study of Flow Patterns Around a Multipurpose Dam.” *Hydroinformatics 2002: Proceedings of the Fifth International Conference on Hydroinformatics*, Cardiff, Wales, Jul 1-5, 2002, R.A. Falconer et al., eds., IWA Publishing, London, UK.