Received: 30 April 2020 | Revised: 14 July 2020 | Accepted: 16 July 2020
DOI: 10.1002/rra.3710

REVIEW PAPER

WILEY

Hybrid modelling of low velocity zones in box culverts to assist fish passage: Why simple is better!

Xinqian Leng[1,2] | Hubert Chanson[2]

[1]I2M, Site ENSCBP, TREFLE, Université de Bordeaux, Pessac, France

[2]School of Civil Engineering, The University of Queensland, Brisbane, QLD, Australia

Correspondence
Xinqian Leng, I2M, Site ENSCBP, TREFLE, Université de Bordeaux, Pessac, France.
Email: xinqian.leng@uqconnect.edu.au

Funding information
Australian Research Council, Grant/Award Number: LP140100225; Department of Transport and Main Roads, Queensland Government, Grant/Award Number: TMTHF 1805; School of Civil Engineering, University of Queensland

Abstract

A culvert is a covered road structure constructed to pass flood and drainage. The engineering design principle of culverts focuses on optimising flood capacity at the lowest possible cost, resulting in high flow velocity through the barrel. With modern engineering design being more environmentally aware, culverts are considered common barriers for fish migrating upstream due to the excessive barrel velocity. The migration of fish upriver is important for their breeding and feeding activities, contributing to a stable population and species diversity. Improved or novel design for a fish-friendly culvert is needed to ensure well-being of local fish community. This article is a review of recent developments in seeking a solution for a better fish-friendly culvert design. A major common feature of the reviewed studies is the application of hybrid modelling, combining laboratory experiments and numerical Computational Fluid Dynamics (CFD) modelling together. Various design alternatives were compared for pros and cons: channel roughening, installation of speed-reducing structures (e.g., baffles or beams), and simply widening the channel by adding more culvert units (boxes). Overall, channel widening stands to be a better design choice due to its suitability for both flood passage and fish passage, with little impact on operation safety and a reduced maintenance cost.

KEYWORDS

baffles, longitudinal rib, culvert, fish passage, hydraulic modelling, hydrodynamics, remediation

1 | INTRODUCTION

Low-level river crossings deliver a range of socio-economic services, in terms of terrestrial connectivity and hydraulic controls. The structures have however some adverse impact on freshwater river system morphology and ecology, including in terms of upstream passage of fish and aquatic life (Anderson et al., 2012; Warren Jr. & Pardew, 1998). Obstructions of fish movement may be caused by a range of situations encompassing perched outlet, high velocities in the throat of the culvert, debris accumulation at the inlet, standing waves in the engineered waterway (Behlke, Kane, McLeen, & Travis, 1991; Olsen & Tullis, 2013). With small weak-swimming fish species and juveniles of larger fish, the high water velocities in the culvert barrel are often a prevailing impediment. For the past three decades, a number of culvert design guidelines were developed to improve the fish traversability of the structures (e.g., Bates, Barnard, Heiner, Klavas, & Powers, 2003; Hunt, Clark, & Tkach, 2012; Kilgore, Bergendahl, & Hotchkiss, 2010), not always successfully, and lacking robust engineering-based methodology (Leng, Chanson, Gordos, & Riches, 2019).

Culverts are common hydraulic structures built under embankments to pass streams and water runoff (Figure 1). There exist a range of culvert types, for example, Figure 1a shows an older masonry structure, Figure 1b presents a pipe culvert, while Figure 1c,d illustrate multicell box culvert structures. The selection depends on a number of factors including discharge capacity requirements, maximum acceptable water level on the upstream floodplain and site geometry (Schall, Thompson, Zerges, Kilgore, & Morris, 2012). During culvert

FIGURE 1 Photographs of standard culverts. (a) Stone masonry culvert outlet along the Flora River, St Alban (France) on December 23, 2019. (b) Pipe culvert in St Lucia QLD (Australia) on February 2, 2020—Inlet operation after 60–65 mm of rainfall in the past 7–8 hours. (c) Box culvert beneath the Townsville-Mt Isa railway line, Australia on November 16, 2019. (d) Multicell box culvert inlet along East Creek, beneath James St and Mary St, Toowoomba (Australia) on March 1, 2020 [Color figure can be viewed at wileyonlinelibrary.com]

operation, the flow through the barrel is turbulent. The complicated nature of turbulence renders the prediction of the flow field difficult. Traditional one-dimensional (1D) and two-dimensional (2D) numerical models are inappropriate. A more advanced modelling technique of the full flow field in a culvert is Computational Fluid Dynamics (CFD) with a three-dimensional (3D) domain. However, the heavy demand in terms of computational power, time and expertise is not always practical in the most common industrial applications (Rodi, 2017; Toombes & Chanson, 2011). Physical modelling may provide realistic physically-meaningful flow features, but well-known limitations include time, cost, human resources, rigidity in changing geometries and flow conditions, technical expertise and scale effects, when conducted in small-size facilities (Muste et al., 2017).

A major challenge in designing fish-friendly culverts is meeting the needs of weak-swimming small-bodied species and juveniles of larger fish, with a characteristic fish swim speed less than 0.6 m/s and a body length less than 150 mm. In New South Wales (Australia), a mean barrel velocity of 0.3 m/s or less was required by the state regulatory authority, a constraint hard to achieve in a culvert operating at design discharge and rarely implemented (DPI-Fisheries, 2013; Leng et al., 2019). In Queensland (Australia), full-height sidewall baffles are required to aid fish passage in culverts (Department of Agriculture and Fisheries, 2018). Other alternatives such as channel roughening and longitudinal beams on sidewalls have been suggested.

This article compares the effectiveness of different velocity-reducing practices, including bed and sidewall roughening, installation of intrusive structures such as sidewall baffles or beams, and simply widening the channel, for example, with additional boxes, and their impacts on the culvert performance and capacity. The work is based upon a composite approach, combining physical modelling performed under controlled laboratory conditions and 3D CFD modelling. Such a method is sometimes called hybrid modelling. A major advantage of hybrid modelling in hydraulic structure design is an optimisation of resources, combining the flexibility of CFD numerical modelling to reduce the material and time costs in building numerous large-scale physical model, while operating large-size physical models to produce realistic boundary conditions, initial conditions and validation data sets for numerical CFD modelling, reducing the total simulation costs.

The composite approach may include loops, feedback and interactions between the physical and CFD numerical techniques, bringing new capabilities to the design of fish-friendly culvert structures.

2 | CURRENT PRACTICES FOR STANDARD BOX CULVERT DESIGNS: A REVIEW

Standard concrete box culverts are the main focus herein, as pipe culverts are considered less suitable for fish passage because of the larger flow velocity and smaller boundary layer region, hence low velocity zones, during operation (Briggs & Galarowicz, 2013; Chanson, 2019a, 2020). From a hydraulic engineering perspective, a culvert is designed to pass the design discharge Q_{des} as a free-surface flow with sufficient free-board, while minimising the afflux and total cost (Chanson, 2004; Herr & Bossy, 1965; Schall et al., 2012). The process leads to a minimisation of the culvert barrel size, associated with fast flowing waters in the barrel at the design discharge.

Current design procedures use the design flow rate Q_{des} and maximum acceptable afflux h_{max} to find the minimum required barrel cross-section area to ensure an inlet control operation (CPAA 2012). The calculations may be performed using theoretical considerations, nomographs or hydraulic design softwares incorporating the culvert design equations. The barrel size is selected through an iterative process (Chanson, 2004; Herr & Bossy, 1965). Assuming inlet control, the minimum barrel width is estimated for the maximum acceptable afflux. The barrel size is then tested for outlet control operation to deduce the afflux under outlet control. If the upstream head is higher for outlet control, additional cells (boxes) must be added and the calculations are repeated, until the number of cells (boxes) ensures inlet control operation, that is, $h_{max}^{inlet} > h_{max}^{outlet}$.

As such, the current design method was developed only for the design flow conditions, usually selected in relation to the local rainfall and runoff data, for example, 20% annual exceedance probability (AEP) storm. For less-than-design flows, the safety of the culvert structure and embankment must be ensured at all times. Potential operational issues encompass downstream damage to the river bed, embankment overtopping, debris blockage, culvert siltation, and barrel misalignment. (Chanson, 2004; Chanson & Leng, 2020). For discharges larger than the design discharge, some erosion and damage may be acceptable but the stability and integrity of the embankment must be ensured. These objectives are achieved by a combination of correct culvert barrel design, as well as a correct design of the inlet and outlet sections to guide the flow into and out of the culvert barrel, including provision of a downstream apron if required.

3 | FISH-FRIENDLY CULVERT DESIGN APPROACHES

3.1 | Presentation

For the design of fish-friendly culverts, a key challenge is to slow down the water flow in the barrel, because excess barrel velocity is a major barrier for upstream fish passage and traversability, in particular for small-bodied native species and juveniles of larger fish. Three recent approaches dedicated to flow velocity reduction in the barrel are covered herein: (a) asymmetrical sidewall roughening, (b) installation of speed-reducing structures such as baffles or streamwise rib, and (c) widening the barrel channel. Each of these approaches presents both strengths and limitations. The following sections use detailed physical and CFD numerical studies to evaluate the pros and cons.

3.2 | Asymmetrical boundary roughening

Boundary roughening induces a decrease in flow velocity by increasing boundary friction and thickening the boundary layer regions. Wang, Chanson, Kern, and Franklin (2016); Wang, Uys, and Chanson (2018) conducted physical experiments to study the effect of various boundary roughness on fish passage. The study included both hydrodynamic and fish behaviour investigations with small-bodied fish, adult Duboulay's rainbowfish (*Melanotaenia duboulay*) and juvenile silver perch (*Bidyanus bidyanus*) (Figures 5c,d). The experiments were performed in a 12 m long 0.5 m wide (0.478 m wide with sidewall roughness) rectangular prismatic channel, acting a 1:1 scale model of a two-lane rural road box culvert barrel. Three roughness configurations were compared: (a) a reference smooth invert, (b) a rough invert with an equivalent sand roughness height ($k_s \sim 20$ mm), and (b) a rough invert and asymmetrical rough sidewall ($k_s \sim 30$ mm). The velocity measurements were conducted using an acoustic Doppler velocimeter (ADV) and a Prantl-Pitot tube. All data showed the marked effect of boundary roughness on the distributions of time-averaged velocity and velocity fluctuations. With the rough bed and rough sidewall (config. 3), the results showed an asymmetrical velocity field, the existence of the velocity dip and the presence of secondary currents (Figure 2). The asymmetrical roughness configuration appeared to provide excellent recirculation regions next to the rough sidewall and at the corner between the rough sidewall and rough bed, suitable to the upstream passage of small-body-mass fish (Wang et al., 2016; Wang & Chanson, 2018).

A numerical study was carried out using the Computational Fluid Dynamics (CFD) package ANSYS™ FLUENT (V. 18.0) to simulate the full velocity field of the asymmetrical roughened barrel (Zhang & Chanson, 2018). The numerical work used a numerical domain of 12 m × 0.5 m × 0.5 m (length×width×height). A 7-equation Reynolds Stress Model (RSM) coupled with a Volume of Fluid (VOF) method was applied, using a hex-dominate structured mesh grid with a maximum of 5,000,000 grid points. Overall the results showed a good agreement between CFD numerical and physical data in terms of free-surface estimation and about 5–10% discrepancies in velocity estimates (Figure 3). The findings demonstrated the ability of CFD models to provide a relatively high-fidelity data set, conducive to a solid understanding of the hydrodynamic mechanisms responsible to improve fish passage in an asymmetrically roughened channel (Zhang & Chanson, 2018). Numerical calculations with several types

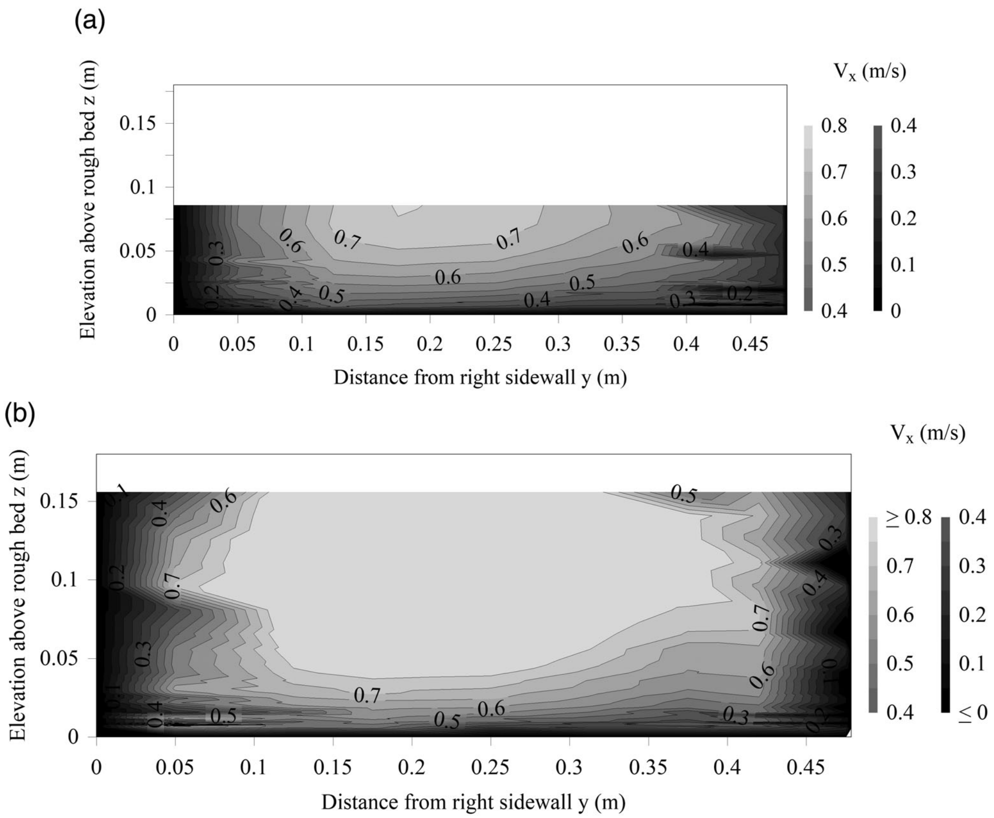

FIGURE 2 Streamwise velocity V_x contour maps in a 12 m long 0.5 m box culvert barrel flume with rough bed and sidewall: physical measurements—Data: (Wang et al., 2016), x = 0 at the upstream inlet; y = 0 at the right smooth sidewall, streamwise velocity scale in m/s, un-distorted axis scale. (a) x = 8 m, Q = 0.0261 m^3/s. (b) x = 8 m, Q = 0.0556 m^3/s

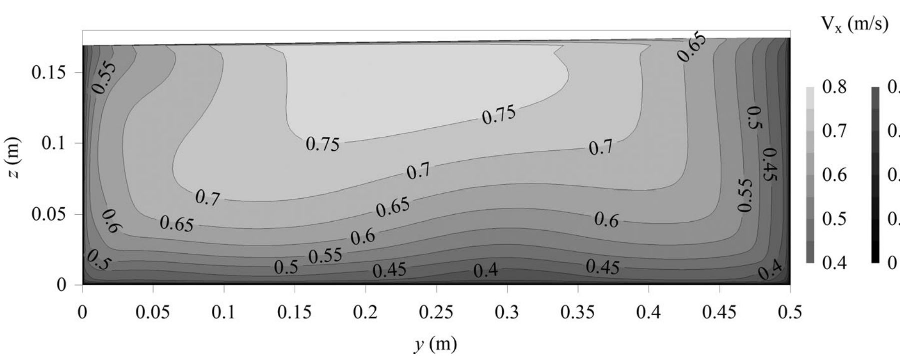

FIGURE 3 Streamwise velocity V_x contour maps in a 12 m long 0.5 m box culvert barrel flume with rough bed and sidewall: CFD numerical experiments—Data: Zhang and Chanson (2018), x = 8 m, Q = 0.0556 m^3/s, x = 0 at the upstream inlet; y = 0 at the right smooth sidewall, streamwise velocity scale in m/s, un-distorted axis scale

of mesh refinements showed some insensitivity of the free-surface level to mesh grid density. The velocity distributions appeared to be more sensitive to the aspect ratio than the grid cell count. The largest discrepancies were observed in terms of the flow resistance, with a difference of 5 to 10% between the two finest meshes.

A key outcome is the ability of the CFD model to simulate secondary flow patterns, matching closely experimental observations as illustrated in Figure 4. Figure 4 presents the velocity vector field in the plane perpendicular to the streamwise direction. The arrows point to five recirculation cells. Further analyses of secondary flow patterns showed a relationship between secondary motions and distribution of wall-tangent Reynolds stresses, indicating that both bed and sidewall boundaries play a role in producing suitable flow regions for upstream fish passage (Chanson, 2019b; Zhang & Chanson, 2018). In summary, the hybrid modelling approach worked successfully for the asymmetrical roughening configuration. Despite its uncertainties and limitation, the CFD numerical modelling can complement the physical modelling, which has its own intrinsic limitation and uncertainties, for example, from instrumentation, lack of geometric flexibility, human resources.

3.3 | Speed-reducing appurtenances

Intrusive appurtenances (e.g., baffles, ribs) are sometimes introduced in culvert and channel flows to slow down the flow, supposedly facilitating upstream fish passage. The installation of such devices needs to deliver some longitudinal connectivity through which fish can traverse. A number of practical problems are faced by engineers, the most significant ones being the reduction of flood capacity and high

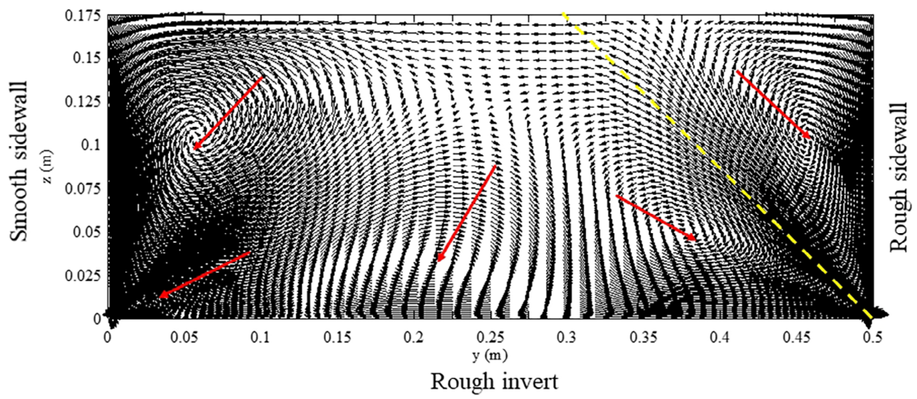

FIGURE 4 Cross-sectional velocity vector field in a 12 m long 0.5 m box culvert barrel flume with rough bed and sidewall: CFD numerical experiments—Data: Zhang and Chanson (2018), x = 5 m, Q = 0.0556 m^3/s, x = 0 at the upstream inlet; y = 0 at the right smooth sidewall, streamwise velocity scale in m/s, undistorted axis scale [Color figure can be viewed at wileyonlinelibrary.com]

risks of blockage caused by the amount of debris and sediments caught by the devices (baffles, ribs). Maintenance and pedestrian safety are other potential issues. In this section, a discussion on the effectiveness of such devices is developed based upon a hybrid modelling approach.

3.3.1 | Baffles

Two types of baffles were recently proposed to aid upstream passage of small-bodied fish in box culvert barrel: (a) small triangular corner baffles and (b) full-height sidewall baffles. The advantage of small triangular baffles compared to larger baffles is the lesser reduction in flood capacity (Cabonce, Fernando, Wang, & Chanson, 2019; Cabonce, Wang, & Chanson, 2018; Chanson & Uys, 2016). Cabonce, Fernando, Wang, and Chanson (2017); Cabonce et al. (2019) conducted detailed physical experiments in a 12 m long 0.5 m wide rectangular prismatic flume acting as a full-scale box culvert barrel, using three different baffle sizes (Figure 5a) and six different longitudinal spacings for each baffle size. The small corner baffles were only installed along one side of the flume (Figure 5). The second type of baffles is full-height sidewall rectangular baffles. Leng and Chanson (Leng & Chanson, 2019; Leng & Chanson, 2020a) conducted detailed physical modelling with three sizes and different longitudinal spacings along one sidewall. With both series of experiments, free-surface elevation measurements were performed, using a pointer gauge (accuracy ±0.5 mm) and velocities were measured by Prandtl-Pitot tube.

Triangular corner baffles

The cross-sectional velocity measurements showed a clear asymmetry of the flow field, with larger velocity skewed towards the smooth unbaffled sidewall (Cabonce et al., 2017, 2019). Typical physical results are presented in Figure 6a. Negative velocity was observed in the near-wake of the baffles, consistent with dye injection observations (Figure 5b), as well as a separation zone downstream of the inclined edge. In front of the baffles, a stagnation zone was observed where fish could rest in a region of low velocity and low turbulence (Figure 5c). Results also highlighted an increase of Low Velocity Zone (LVZ) area, being two to three times larger than the LVZ size for a smooth channel under the same flow conditions.

Importantly, the low velocity zones must be well-connected to ensure that the fish could traverse the entire culvert barrel. In turn, the full flow field must be resolved at every location between two adjacent baffles, which is only practically feasible using CFD numerical simulation. Zhang and Chanson (2018) performed CFD simulations with two corner baffle sizes, that is, h_b = 0.067 m and 0.133 m, and one longitudinal spacing L_b = 0.67 m. A Large Eddy Simulation (LES) was selected because of the flow separation. To save computation time, the numerical model domain only contained the flow section 0.5 × L_b upstream and downstream of one triangular baffle, that is, 0.67 m long × 0.5 m wide with the height equal to the flow depth measured experimentally, using periodic boundaries for inlet and outlet, and a symmetry condition for the top boundary. The mesh grid was a structured grid with hexagon-dominate mesh, consisting of maximum 567,517 grid points. A typical result is shown in Figure 6b.

The CFD results showed that the small corner baffles are effective in creating low velocity zones immediately behind the baffle, but these zones could lack a good longitudinal connectivity depending upon the water discharge and baffle configuration (size and spacing). In addition, the negative velocities and recirculation zone could disorient small fish (Cabonce et al., 2018). A similar scenario was observed physically. Overall, the numerical results showed the CFD modelling capacity to satisfactorily replicate major hydrodynamic features, such as separation and secondary flow structures, despite a relatively simple coarse mesh. A qualitative perspective needs to be adopted when applying these results as it is still challenging to provide quantitatively accurate velocity field everywhere. Discrepancies may result from a number of factors including boundary treatment, mesh density and quality, and types of simulation (transient or steady). With the complexity of flow description and computational resource constraints combined, similar numerical models should be applied as a qualitative assistance rather than predictive measure for optimising the design of baffle systems (Zhang & Chanson, 2018).

Full-height sidewall baffles

Physical modelling of full-height sidewall rectangular baffles was conducted with three baffle sizes, scaled based upon the

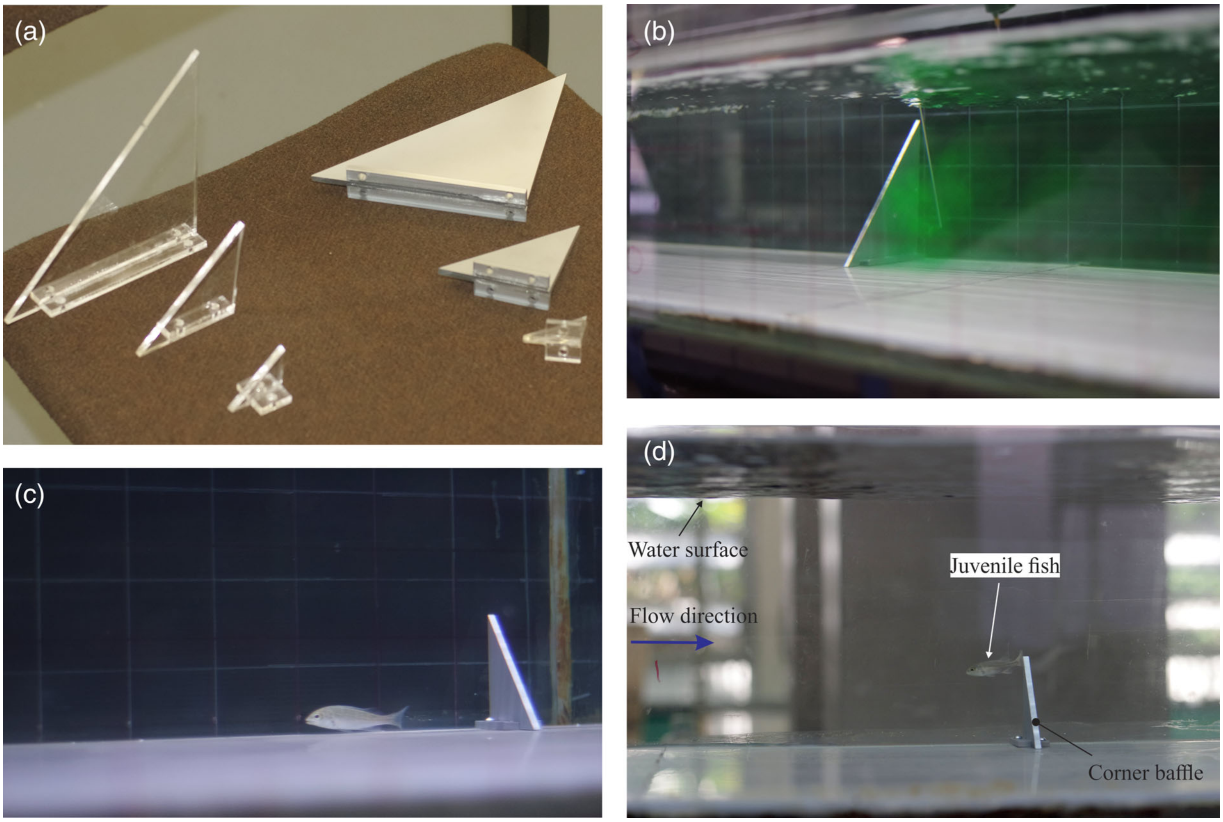

FIGURE 5 Small corner baffles in a 12 m long 0.5 m box culvert barrel flume: physical experiments. (a) triangular corner baffles of three sizes: h_b = 0.033 m, 0.066 m and 0.133 m from foreground to background. (b) Flow recirculation behind a small corner baffle visualised with green dye injection—Flow direction from left to right, Q = 0.035 m^3/s, h_b = 0.133 m, L_b = 0.667 m (left sidewall corner only). (c) Juvenile Silver perch (*Bidyanus bidyanus*) resting in the stagnation zone upstream of a triangular corner baffle - Flow direction from left to right, Q - 0.0556 m^3/s, h_b = 0.067 m, L_b = 0.667 m (left sidewall corner only). (d) Juvenile Silver perch (*Bidyanus bidyanus*) negotiating the upstream progression past a triangular corner baffle - Flow direction from left to right, Q - 0.0556 m^3/s, h_b = 0.067 m, L_b = 0.667 m (left sidewall corner only) [Color figure can be viewed at wileyonlinelibrary.com]

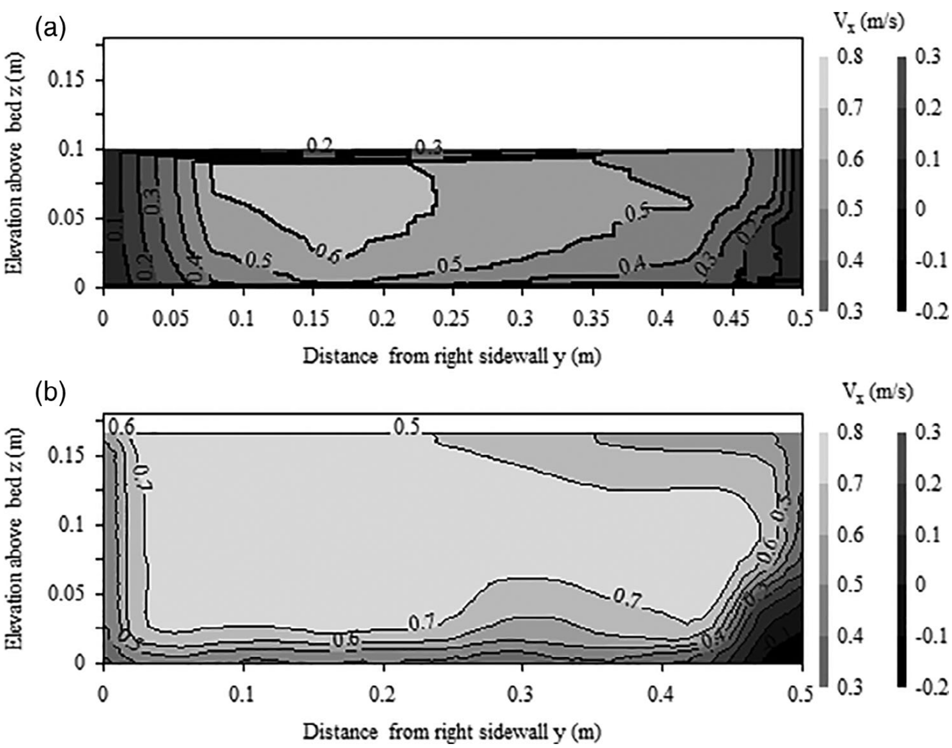

FIGURE 6 Streamwise velocity V_x contour maps in a 12 m long 0.5 m box culvert barrel flume with small triangular baffles (left sidewall corner only): physical and CFD numerical data—Q = 0.0556 m^3/s, h_b = 0.067 m, L_b = 0.667 m, $(x\text{-}x_b)/L_b$ = 0.25; $x\text{-}x_b$ = 0 at the baffle, y = 0 at the right smooth sidewall, streamwise velocity scale in m/s, un-distorted axis scale. (a) Physical measurements—Data: (Cabonce et al., 2017). (b) CFD numerical results—Data: (Zhang and Chanson 2018)

recommendation from DAF (2018) (Figure 7). The numerical CFD modelling is conducted in parallel by Transportation and Main Roads (TMR) Queensland. Herein, only the physical modelling results are presented for a preliminary view on this design alternative.

The physical experiments were performed on a 12 m long 0.5 m wide rectangular prismatic channel with a false bed on which the baffles are fixed. Only one side of the channel was baffled (Figure 7). Visual observations, free-surface, and velocity measurements were performed using a high-speed high-definition video camera (Casio™ Exilim EX10), point gauge and Prantl-Pitot tube, respectively. The experimental data delivered a fine characterisation of the hydrodynamics of the asymmetrical baffled culvert barrel. A typical velocity contour is presented in Figure 8. The data showed a very-significant impact of the full-height sidewall baffles on the turbulent flow conditions in the culvert barrel. The observations indicated in particular a substantial increase in flow turbulence and flow resistance, as well as an asymmetrical turbulent velocity field. The study demonstrated a massive reduction in discharge capacity of box culverts (more than 30%) in presence of full-height sidewall baffles for a given design afflux, with an increasing impact with increasing discharge for all baffle configurations (Leng & Chanson, 2020a).

3.3.2 | Streamwise sidewall rib

Recent biological tests suggested that a streamwise sidewall rib might facilitate the upstream passage of small body-mass fish species (Watson, Goodrich, Cramp, Gordos, & Franklin, 2018). Sanchez, Leng, and Chanson (2018); Sanchez, Leng, Brandis-Martini, and Chanson (2020) physically tested the design in an asymmetrical rectangular channel equipped with a 50 mm × 50 mm streamwise rib along the right sidewall, through the entire test section (Figure 9). Both flow visualisations and flow resistance data showed three-dimensional velocity patterns and substantial energy dissipation associated in presence of the longitudinal beam. Strong secondary circulation was observed in the bottom corner cavity beneath the rib (Figure 9b). The total flow resistance was larger than basic skin friction, causing a 30% reduction in the channel discharge capacity for a given afflux. In the ribbed channel, complicated secondary currents of Prandtl's second kind developed, linked to low-velocity and high-turbulence in the bottom corner cavity. The sidewall rib and channel asymmetry contributed to the development of strong secondary motion, and associated turbulent dissipation. A key feature of the design was the provision of a well-marked highly-turbulent low-velocity zone beneath the sidewall rib for all tested flow conditions (Sanchez et al., 2020). The overall slow region areas were cumulatively and relatively small (Figure 10).

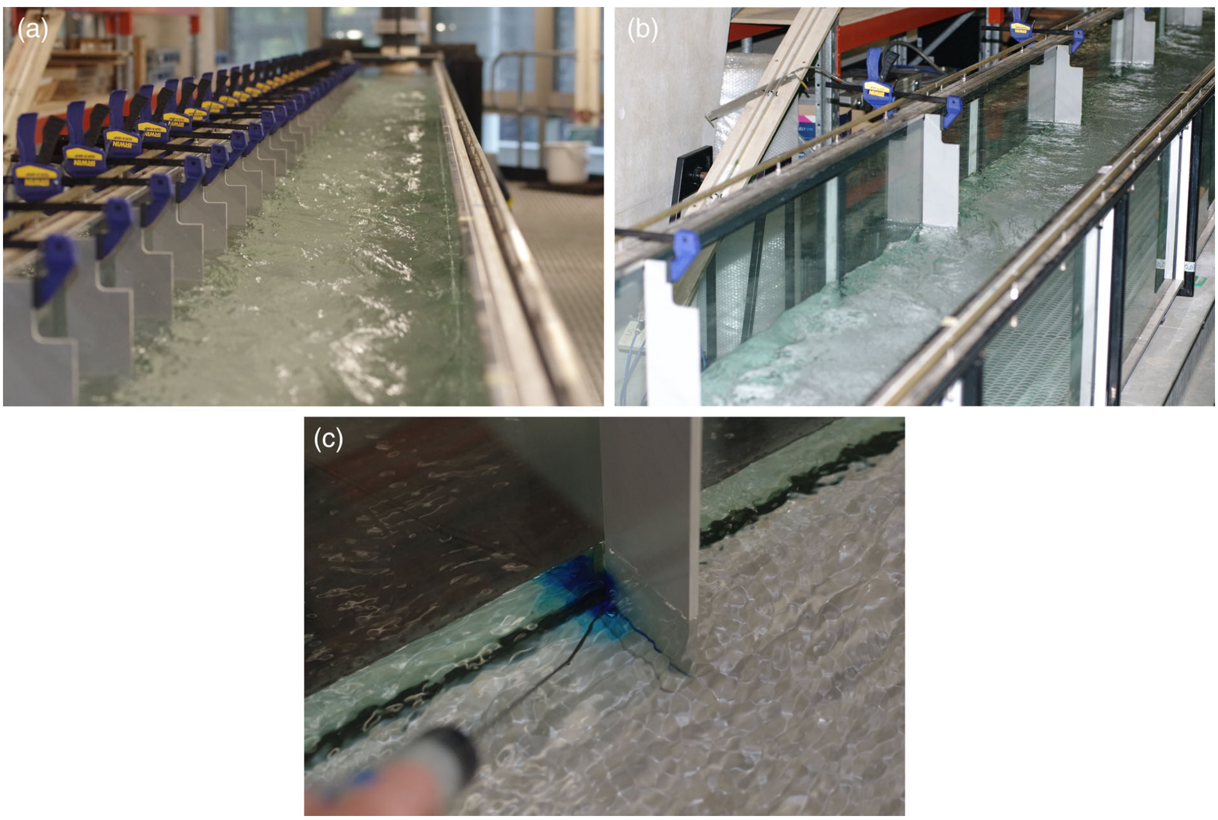

FIGURE 7 Full height sidewall baffles in a 12 m long 0.5 m box culvert barrel flume: physical experiments. (a) Looking upstream, Q = 0.100 m^3/s, h_b = 0.0833 m, L_b = 0.33 m (right sidewall only). (b) Looking upstream, Q = 0.0555 m^3/s, h_b = 0.083 m, L_b = 1.33 m (right sidewall only). (c) Flow recirculation behind a full-height sidewall baffle visualised with blue dye injection—Flow direction from right to left, Q = 0.017 m^3/s, h_b = 0.083 m, L_b = 0.33 m (right sidewall only) [Color figure can be viewed at wileyonlinelibrary.com]

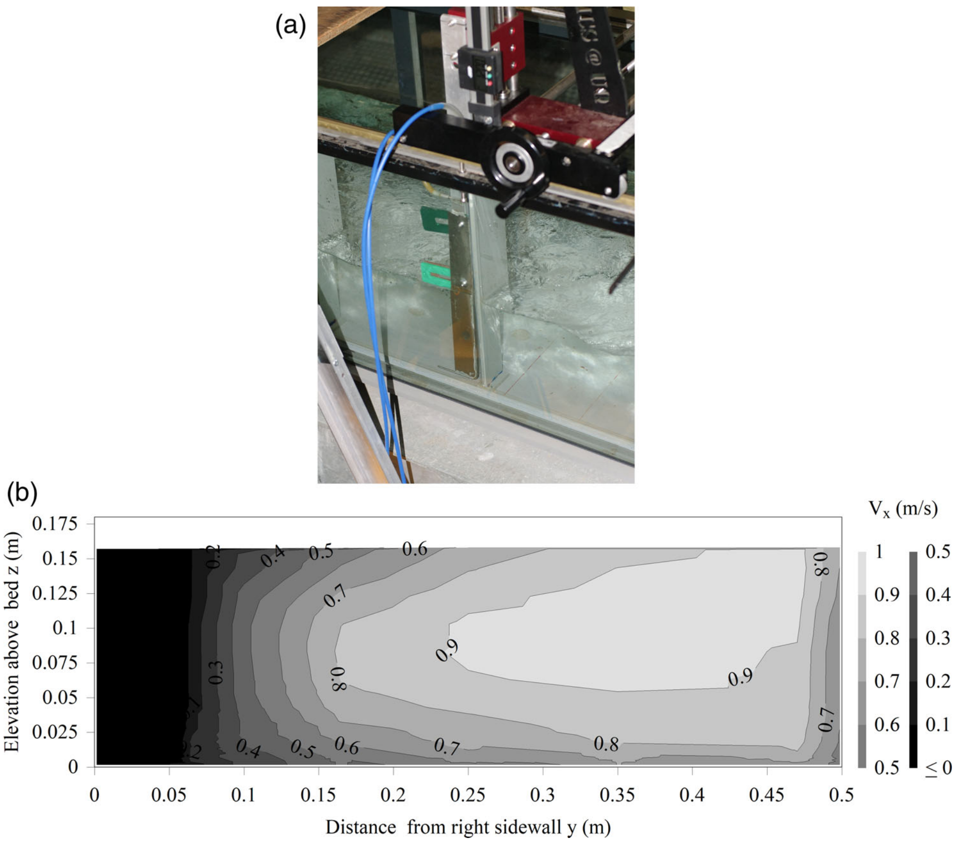

FIGURE 8 Streamwise velocity V_x contour maps in a 12 m long 0.5 m box culvert barrel flume with full-height sidewall baffles: physical measurements - Data: Leng and Chanson (2019), Q = 0.054 m^3/s, h_b = 0.083 m, L_b = 0.33 m (right sidewall only); $(x-x_b)/L_b = 0$ at the baffle; y = 0 at the right smooth sidewall, streamwise velocity scale in m/s, un-distorted axis scale.
(a) Photograph of the flow looking through the right sidewall - Flow direction from left to right.
(b) Contour map at $(x-x_b)/L_b = 0.5$ [Color figure can be viewed at wileyonlinelibrary.com]

A numerical study was carried out using ANSYS™ FLUENT (V. 18.0), applying two types of turbulence models belonging to the RANS family: (a) a standard k-ϵ model and (b) Reynolds Stress Model (RSM). The numerical domain (12 m × 0.5 m × 0.5 m) was discretised into hex-dominate unstructured mesh with spatial refinement near and around the rib. The maximum grid density consisted of 375,717 nodes and 342,550 elements. Both CFD models showed a satisfactory capability to simulate the three-dimensional flow around a longitudinal rib, using a standard k-ϵ or Reynolds Stress model, with the latter being more accurate in simulating the flow motion beneath the rib (Sanchez et al., 2020). All CFD models tended to over-estimate the water velocities in the middle of the channel, and were overall associated with smaller low-velocity zones as a result.

While the streamwise sidewall rib configuration might be applied to hydraulic structures, for example, box culvert barrel, the implications must be considered with uttermost care. A number of practical considerations showed major technical concerns. The rib sharp edges created discontinuities, contributing to enhanced secondary current motion in the vicinity of the rib. Sharp edged external and internal corners played a key role in the tested design. The results were obtained for carefully-aligned square cavity. Any mis-alignment of the rib or rounding of the rib edges might impact adversely onto the hydrodynamics, including in terms of the capability of the system, for example, in terms of heat transfer, upstream fish passage, etc. The siltation and sedimentation of the cavity below the rib is another major operational issue, in sediment-laden systems and natural streams. The sidewall rib is further only suitable to straight channel. In most instances, alternative designs should be preferred (Sanchez et al., 2018, 2020).

3.4 | Barrel channel widening

A simple way to decelerate the barrel velocities without impacting on the culvert discharge capacity and afflux at design flow conditions, or cause further maintenance issue, is to widen the barrel, for example, with additional boxes in a multicell culvert structure. This approach is simple, technically sound and should not be overlooked because of its simplicity. Indeed it may provide the easiest and most cost-efficient method to solve the crisis of fish passage. Some design guideline were developed in terms of hydraulic engineering of box culverts to assist the upstream passage of small-bodied fish and juvenile of large fish. The principal ideas are (a) the box culvert design is prioritised for flood passage at design discharge Q_{des}; and (b) fish passage is achieved for discharges up to a fraction of the design flow Q_T, for example, with $Q_T = 0.1 \times Q_{des}$. Fish passage provision relies the provision of sizeable low velocity zones (LVZa) in each culvert barrel box, mostly along the sidewalls and in the bottom corners. Figure 11 provides a typical flow chart with the steps applied to a fish-friendly culvert design prioritising low velocity zones at less-than-design flows. To quantify the size and dimensions of these LVZs, hence to determine if they are sufficient for fish passage, a hybrid modelling approach was applied.

The physical modelling combined several datasets for the velocity fields on a smooth invert for a 12 m long 0.5 m wide box culvert

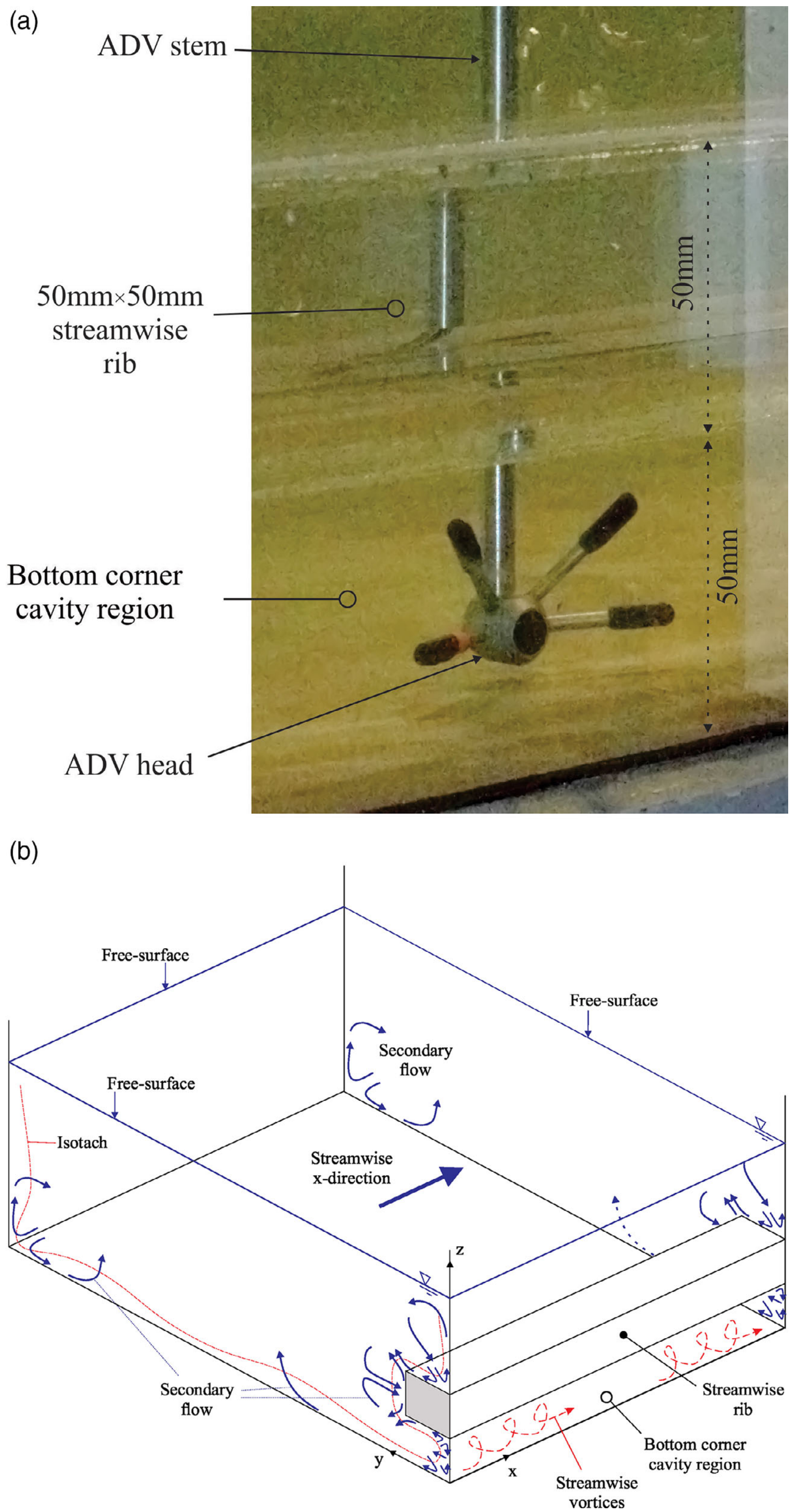

FIGURE 9 Physical modelling of a 12 m long 0.5 m box culvert barrel equipped with a 50 mm × 50 mm streamwise rib with sharp-edged corners. (a) Sideview through the right sidewall - Flow conditions: Q = 0.0556 m^3/s, flow direction from left to right. Note the ADV stem beneath the 50 mm × 50 mm streamwise rib. (b) Sketch of streamlines, recirculation and secondary flows in the asymmetrical box culvert barrel flume (Sanchez et al., 2020) [Color figure can be viewed at wileyonlinelibrary.com]

barrel flume (Cabonce et al., 2017, 2019; Wang et al., 2016, 2018). The experimental conditions were limited to only one box geometry and a few inflow conditions. To develop design guidelines for practical engineering applications, a larger combination of culvert geometry and inflow conditions had been tested. This was undertaken based upon CFD numerical modelling, conducted over a range of inflow conditions, tailwater depth and culvert barrel geometries (Leng & Chanson, 2018).

A key output of CFD modelling was the percentage of flow area under a given velocity, that is, the relative low velocity zone, shown

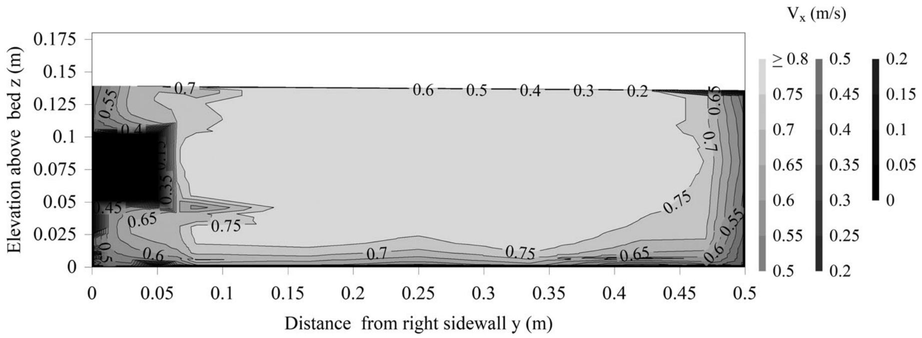

FIGURE 10 Streamwise velocity V_x contour maps in a 12 m long 0.5 m box culvert barrel flume with a streamwise rib: physical measurements—Data: (Sanchenz et al., 2020), Q = 0.0556 m^3/s, x = 8, at the baffle; y = 0 at the right smooth sidewall, 50 mm × 50 mm streamwise rib along right sidewall, velocity scale in m/s, un-distorted axis scale

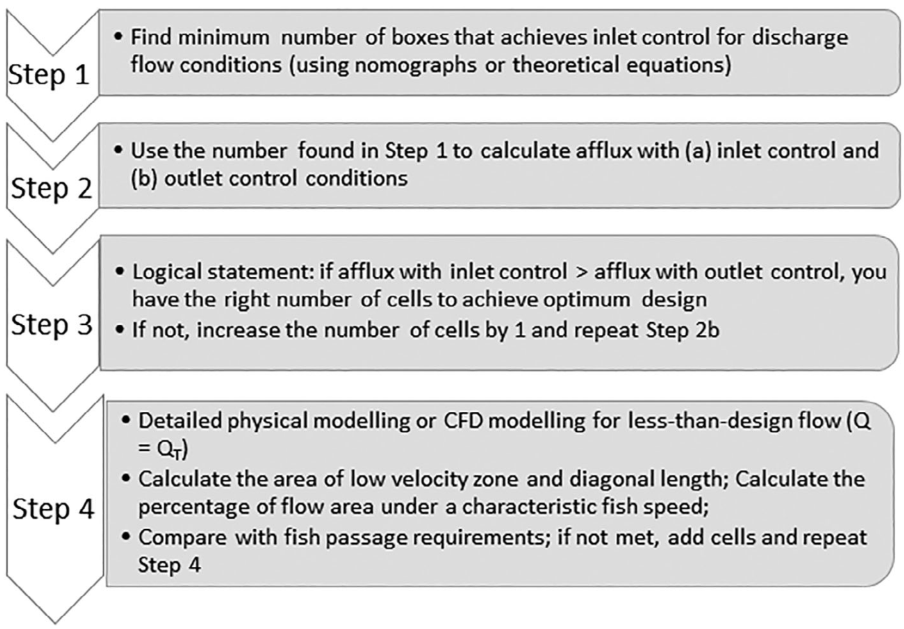

FIGURE 11 Design chart for optimisation of fish friendly smooth box culverts

by Figure 12. The compilation of all physical and numerical cases, including past experimental and CFD data (Leng & Chanson, 2020b), showed a common trend. That is, the relative percentage of flow area under a given velocity increased monotonically with the relative targeted velocity, irrespective of the aspect ratio (i.e., relative channel width). A robust relationship was derived, describing the relationship between the percentage of flow area under a certain relative velocity, normalised to the bulk velocity V_{mean}:

$$A = 100^{1-N} \times \left(\frac{N}{N+1}\right)^N \times \left(\frac{V_x}{V_{mean}}\right)^N \tag{1}$$

Other than area percentage, it is also required that the absolute area of the low velocity zone accommodates at least one fish. Hence additional criteria are imposed such that a minimum area, for example, 25 mm by 25 mm (width by height), of low velocity zone is satisfied at both bottom corners of a barrel cell. The suggested size was proposed based upon the dimensions of small-body-mass Australian native fish species. Larger dimensions may be required for larger fish species, which might also be stronger swimmers thus requiring a lesser stringent barrel velocity to traverse.

4 | DISCUSSION

In recent years, an increased environmental awareness provided new challenges to the design and construction of hydraulic structures. Upstream migration of small-bodied fish and juveniles of larger fish has always been a challenge to embed in the engineering design of culverts, because of a combination of factors: (a) the "conventional" design principle in optimising flood capacity at minimal economic cost; (b) an absence of standardised test methods in fish swim tests, that is, two different studies rarely use the same protocol (Katopodis & Gervais, 2016; Wang & Chanson, 2018); and (c) a lack of engineering perspective in many requirements and standards set by some biological regulatory authorities. The studies reviewed herein took a course of six years, testing a number of alternatives for assisting upstream passage of small-bodied fish in box culverts. Although actual field tests are yet to be done, the results provided a broad scope in which both qualitative and quantitative comparisons were achieved.

Under the consideration of optimising flood capacity and structural safety, it is clear that the culvert barrel widening and smart

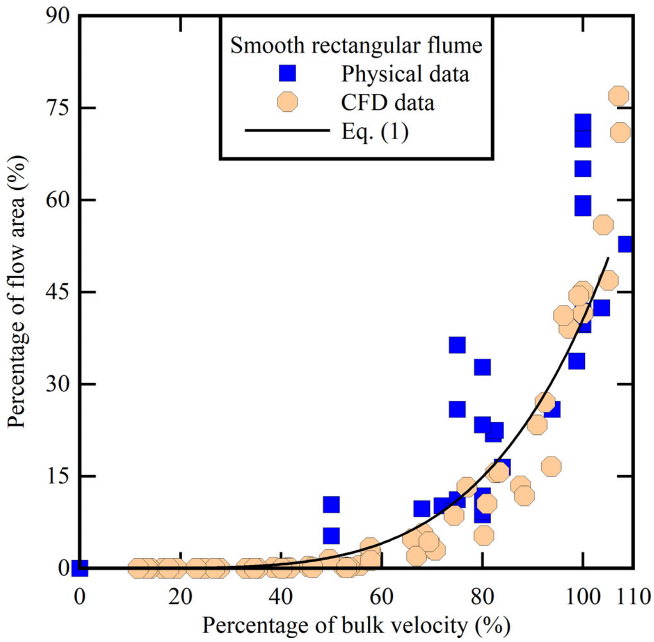

FIGURE 12 Dimensionless area fraction where the velocity is less than a relative streamwise velocity V_x/V_{mean} in percentage (%). Comparison between physical and CFD numerical data [Color figure can be viewed at wileyonlinelibrary.com]

installation of additional boxes would be the best practice (Chanson & Leng, 2020; Leng et al., 2019). While it is seemingly an expensive option, the subsequent reduction of upstream afflux, ease of construction using existing precast units and simplified maintenance play a key role in balancing the total costs. In terms of maximising low velocity zones and good connectivity for fish traversability, the longitudinal rib and full-height rectangular baffles may be "good" options. However, both appurtenance types induce a massive reduction in discharge capacity, that is, up to 30–50%, which in turn raise the total cost of the overall culvert structure, while they pose significant maintenance issue for future operations, that is, debris blockage, siltation, pedestrian safety. In the end, a smart choice is needed. Often the use of complicated structures may not be the best solution: simpler is better!

The importance of hybrid modelling has been illustrated through a series of detailed physical and CFD numerical modelling. Hybrid modelling is not only the validation of numerical models against physical ones, although systematic validation is essential for application of numerical and CFD results. The key feature of hybrid modelling is the combined use of both physical and numerical CFD models as a complement of each other, to make up for the intrinsic limitations of the other modelling technique. In the example of small triangular corner baffles, it was physically impossible to measure the velocity distributions at an infinitesimal interval between two adjacent baffles; CFD modelling was performed to determine the connectivity of low velocity zones between two baffles. This can only be used when the CFD models are validated in-depth against high-quality physical data for the same geometry and inflow conditions, and against the same flow characteristics. In the present application, the time-averaged streamwise velocity component is uppermost important and the CFD validation must include a thorough validation in terms of both free-surface elevations, longitudinal velocity and low velocity zone area. It is worth noting that quantitative discrepancies may still exist between physical and numerical models, even though a high-quality mesh was used and mesh convergence was well achieved. In addition, there is no guarantee that a turbulence modelling (e.g., k-ε, RSM) that previously worked on a set of flow conditions will be suitable for a different set of flow conditions, mainly due to the many empirical constants in the turbulence models. Nonetheless, hybrid modelling could be a time-efficient and physically sound way of designing contemporary hydraulic structures, taking advantage of the pros and complement the cons of both physical and numerical modelling taken separately.

5 | CONCLUSIONS

Hybrid modelling is a scientific engineering approach combining CFD and physical modelling, and can be efficient and effective in modern designs of hydraulic structures. The physical modelling component does not only provide systematic validation data sets but also yields physically sensible initial and boundary conditions for CFD. The numerical CFD modelling component, on the other hand, delivers detailed velocity and turbulence field information throughout the entire test domain, complementing the physical results. The current study of finding a better practice for fish-friendly culvert design, presented herein, is an example of successful application of hybrid modelling.

In terms of the design alternatives, the present review detailed a list of the most recent research works, and the key conclusions through comparing and contrasting are:

1. The smooth box culvert widening and associated design guidelines are the easiest alternative to implement and are best suited for the primary purpose of a box culvert structure which is flood passage.
2. The key advantage of culvert widening is its general approach which can be applied to a broad range of fish species, adults and juveniles. Yet field testing is needed to verify the effectiveness of the methodology on real fish behaviour and passage.
3. Full-height rectangular baffles and longitudinal beam could both provide continuous low velocity zones for fish traversability.
4. The downsides of all large intrusive bodies are that the impact of relatively high turbulence on fish behaviour remain poorly understood, especially at medium to large discharges. The installation and maintenance of both types of intrusive appurtenances can also be difficult and operationally challenging.
5. All large intrusive appurtenances reduce the flood capacity by at least 30% and sometimes much more.

Overall, the current review illustrated the importance of "smart" engineering design of hydraulic structures in a complicated modern design scenario: simple is better.

ACKNOWLEDGMENTS

The authors thank the many people who provided valuable inputs, exchanges and discussions, in particular Professor Colin J. APELT (The University of Queensland, Australia), Dr Matthew GORDOS and Marcus RICHES (NSW DPI Fisheries), and Dr Hang WANG (Sichuan University, China).

CONFLICT OF INTRESTS

Hubert CHANSON has competing interest and conflict of interest with a Craig E. FRANKLIN.

DATA AVAILABILITY STATEMENT

Data available on request from the authors: The data that support the findings of this study are available from the corresponding author upon reasonable request.

ORCID

Xinqian Leng https://orcid.org/0000-0001-8472-7925
Hubert Chanson https://orcid.org/0000-0002-2016-9650

REFERENCES

Anderson, G. B., Freeman, M. C., Freeman, B. J., Straight, C. A., Hagler, M. M., & Peterson, J. T. (2012). Dealing with uncertainty when assessing fish passage through culvert road crossings. *Environmental Management*, *50*, 462–477.

Bates, K., Barnard, B., Heiner, B., Klavas, J. P., & Powers, P. D. (2003). In Washington Department of Fish and Wildlife (Ed.), *Design of road culverts for fish passage* (Vol. 111). Olympia, Washington, DC: Washington Department of Fish and Wildlife.

Behlke, C.E., Kane, D.L., McLeen, R.F., & Travis, M.T. (1991). "Fundamentals of culvert design for passage of weak-swimming fish." *Report FHW A-AK-RD-90-10*, Department of Transportation and Public Facilities, State of Alaska, Fairbanks, USA, 178 pages.

Briggs, A. S., & Galarowicz, T. L. (2013). Fish passage through culverts in central Michigan warmwater streams. *North American Journal of Fisheries Management*, *33*, 652–664.

Cabonce, J., Fernando, R., Wang, H., & Chanson, H. (2017). "Using triangular baffles to facilitate upstream fish passage in box culverts: Physical modelling." Hydraulic Model Report No. CH107/17, School of Civil Engineering, The University of Queensland, Brisbane, Australia, 130 pages (ISBN 978–1-74272-186-6).

Cabonce, J., Fernando, R., Wang, H., & Chanson, H. (2019). Using small triangular baffles to facilitate upstream fish passage in standard box culverts. *Environmental Fluid Mechanics*, *19*(1), 157–179. https://doi.org/10.1007/s10652-018-9604-x

Cabonce, J., Wang, H., & Chanson, H. (2018). Ventilated corner baffles to assist upstream passage of small-bodied fish in box culverts. *Journal of Irrigation and Drainage Engineering*, *144*(8), 04018020. https://doi.org/10.1061/(ASCE)IR.1943-4774.0001329

Chanson, H. (2004). The hydraulics of open channel flow: An introduction. In *Butterworth-Heinemann* (Vol. 630, 2nd ed.). Oxford, UK.Butterworth-Heinemann.

Chanson, H. (2019a). "Physical modelling of semi-circular channels and low velocity zones—application to pipe culverts and upstream fish passage at less-than-design flows." *Hydraulic Model Report No. CH116/19*, School of Civil Engineering, the University of Queensland, Brisbane, Australia, 101 pages.

Chanson, H. (2019b). Utilising the boundary layer to help restore the connectivity of fish habitats and populations. An engineering discussion. *Ecological Engineering*, *141*, 105613.

Chanson, H. (2020). Low velocity zone in smooth pipe culvert with and without streamwise rib for fish passage. *Journal of Hydraulic Engineering*, *146*, 04020059.

Chanson, H., & Leng, X. (2020). *Fish swimming in turbulent waters. Hydraulics guidelines to assist upstream fish passage in box culverts*. Leiden, The Netherlands: *CRC Press*. https://doi.org/10.1201/9781003029694

Chanson, H., & Uys, W. (2016).Baffle designs to facilitate fish passage in box culverts: A preliminary study. Paper presented at: Proceedings of 6th IAHR International Symposium on Hydraulic Structures, Hydraulic Structures and Water System Management, B. Crookston & B. Tullis Editors, 27–30 June, Portland OR, pp. 295–304.

DAF. (2018). Accepted development requirements for operational work that is constructing or raising waterway barrier works. *Fisheries Queensland*, *77*, 1–77.

DPI-Fisheries. (2013). *Policy and guidelines for fish habitat conservation and management* (p. 80). Wollongbar, NSW, Australia: *NSW Department of Primary Industries*.

Herr, L. A., & Bossy, H.G. (1965). "Hydraulic charts for the selection of highway culverts." *Hydraulic Eng. Circular*, US Dept. of Transportation, Federal Highway Admin., HEC No. 5, December, 50 pages.

Hunt, M., Clark, S., & Tkach, R. (2012). Velocity distributions near the inlet of corrugated steep pipe culverts. *Canadian Journal of Civil Engineering*, *39*, 1243–1251.

Katopodis, C., & Gervais, R. (2016). "Fish swimming performance database and analyses." *DFO CSAS Research Document No. 2016/002*, Canadian Science Advisory Secretariat, Fisheries and Oceans Canada, Ottawa, Canada, 550 pages.

Kilgore, R.T., Bergendahl, B.S., & Hotchkiss, R.H. (2010). Culvert Design for Aquatic Passage. *Hydraulic Engineering Circular Number No. 26*, Federal Highway Administration publication no. FHWA-HIF-11-008, 234 pages.

Leng, X., & Chanson, H. (2020a). Full-height sidewall baffles in box culvert to assist upstream fish passage: Physical modelling. Paper presented at: International Symposium on Hydraulic Structures (ISHS2020) *Proceedings of the 8th IAHR International Symposium on Hydraulic Structures (ISHS2020)*, 12–15 May 2020, Santiago, Chile, R. JANSSEN and H. CHANSON Editors, The University of Queensland, Brisbane, Australia, 10 pages (Accepted, In print)

Leng, X., & Chanson, H. (2018). Modelling low velocity zones in box culverts to assist fish passage. Paper presented at: Proceedings of 21st Australasian Fluid Mechanics Conference, Adelaide, Australia, 10–13 December, editors T.C.W. LAU and R. M. KELSO, paper 547, 4 pages.

Leng, X., & Chanson, H. (2019). Physical modelling of sidewall baffles in standard box culvert barrel to assist upstream fish passage. *Hydraulic Model Report No. CH115/19*, School of Civil Engineering, The University of Queensland, Brisbane, Australia, 87 pages & 8 movies.

Leng, X., & Chanson, H. (2020b). Hybrid modelling of low velocity zones in box culverts to assist upstream fish passage. *Environmental Fluid Mechanics*, *20*(2), 415–432. https://doi.org/10.1007/s10652-019-09700-1

Leng, X., Chanson, H., Gordos, M., & Riches, M. (2019). Developing cost-effective design guidelines for fish-friendly box culverts, with a focus on small fish. *Environmental Management*, *63*(6), 747–758. https://doi.org/10.1007/s00267-019-01167-6

Olsen, A., & Tullis, B. (2013). Laboratory study of fish passage and discharge capacity in slip-lined, baffled culverts. *Journal of Hydraulic Engineering*, *139*(4), 424–432.

Muste, M., Lyn, D. A., Admiraal, D., Ettema, R., Nikora, V., & Garcia, M. H. (2017). Experimental Hydraulics: Methods, Instrumentation, Data Processing and Management: Volume I: Fundamentals and Methods. IAHR monograph, CRC Press, 496 pages.

Rodi, W. (2017). Turbulence modeling and simulation in hydraulics: A historical review. *Journal of Hydraulic Engineering*, *143*(5), 03117001. https://doi.org/10.1061/(ASCE)HY.1943-7900.0001288

Sanchez, P.X., Leng, X., Von Brandis-Martini, J., & Chanson, H. (2020). Hybrid modelling of low velocity zones in an asymmetrical channel with sidewall longitudinal rib to assist fish passage. *River Research and Applications*, *36*(5), 807–818. http://dx.doi.org/10.1002/rra.3600.

Sanchez, P.X., Leng, X., & Chanson, H. (2018). Fluid dynamics and secondary currents in an asymmetrical rectangular canal with sidewall streamwise Rib. *Hydraulic Model Report No. CH113/18*, School of Civil Engineering, The University of Queensland, Brisbane, Australia, 158 pages (ISBN 978-1-74272-223-8).

Schall, J.D., Thompson, P.L., Zerges, S.M., Kilgore, R.T., & Morris, J.L. (2012). Hydraulic design of highway culverts. *Hydraulic Design Series No. 5*, Federal Highway Administration, Publication No. FHWA-HIF-12-026 3rd edition, 326 pages.

Toombes, L., & Chanson, H. (2011). Numerical limitations of hydraulic models. Proc. 34th IAHR World Congress, Brisbane, Australia, 26 June-1 July, Engineers Australia Publication, Eric VALENTINE, Colin APELT, James BALL, Hubert CHANSON, Ron COX, Rob ETTEMA, George KUCZERA, Martin LAMBERT, Bruce MELVILLE and Jane SARGISON Editors, pp. 2322-2328VALLENTINE, H.R. (1969). "Applied Hydrodynamics." Butterworths, London, UK, SI edition.

Wang, H., & Chanson, H. (2018). On upstream fish passage in standard box culverts: Interactions between fish and turbulence. *Journal of Ecohydraulics, IAHR*, *3*(1), 18–29. https://doi.org/10.1080/24705357.2018.1440183

Wang, H., Chanson, H., Kern, P., & Franklin, C. (2016). "Culvert hydrodynamics to enhance upstream fish passage: Fish response to turbulence." P*roceedings of 20th Australasian Fluid Mechanics Conference*, Australasian fluid mechanics society, G. IVEY, T. ZHOU, N. JONES, S. DRAPER editors, Perth WA, Australia, 5–8 December, paper 682, 4 pages.

Wang, H., Uys, W., & Chanson, H. (2018). Alternative mitigation measures for fish passage in standard box culverts: Physical modelling. *Journal of Hydro-environment Research, IAHR*, *19*, 214–223. https://doi.org/10.1016/j.jher.2017.03.001

Warren, M. L., Jr., & Pardew, M. G. (1998). Road crossings as barriers to small-stream fish movement. *Transactions of the American Fisheries Society*, *127*, 637–644.

Watson, J., Goodrich, H., Cramp, R., Gordos, M., & Franklin, C. (2018). Utilising the boundary layer to help restore the connectivity of fish habitat and population. *Ecological Engineering*, *122*, 286–294.

Zhang, G., & Chanson, H. (2018). Three-dimensional numerical simulations of smooth, asymmetrically roughened, and baffled culverts for upstream passage of small-bodied fish. *River Research and Applications*, *34*(8), 957–964. https://doi.org/10.1002/rra.3346

How to cite this article: Leng X, Chanson H. Hybrid modelling of low velocity zones in box culverts to assist fish passage: Why simple is better!. *River Res Applic*. 2020;36:1765–1777. https://doi.org/10.1002/rra.3710