

Article

Improving Flows in Misaligned Culverts

Rick Jaeger ¹, Carolyn Jacobs ^{1,*} , Katharina Tondera ² and Neil Tindale ¹

¹ School of Science and Engineering, University of the Sunshine Coast, Sippy Downs, QLD 4556, Australia; rjaeger@usc.edu.au (R.J.); ntindale@usc.edu.au (N.T.)

² IMT Atlantique Bretagne—Pays de Loire, Dept. of Energy Systems and Environment, 44307 Nantes, France; ktondera@usc.edu.au

* Correspondence: cjacobs1@usc.edu.au

Received: 24 July 2019; Accepted: 9 September 2019; Published: 16 September 2019



Abstract: This study investigated different approaches to optimize flows in misaligned culverts. Structures aligned with the natural stream are always preferred, as misalignments cause a change of direction at the culvert inlet associated with lower performance and sedimentation and erosion problems. This optimal positioning can cause high financial costs and a flow optimization minimizing the associated problems could be a viable alternative. In this study, we used computational fluid dynamics analysis to evaluate the flow in 44 different scenarios with misalignment angles ranging from 0° to 90°. It was found that smooth transitions towards the narrowest point in the stream (culvert) were possible for any degree of misalignment resulting in improved, uniform velocity distributions and less turbulence. An experimental setup was able to confirm the possible flow improvements. The proposed approach of flow redirection can lower construction costs and gives planners and designers more flexibility as tailored reinforcement and redesign of the stream embankment can be used as an alternative to costly creek alignments.

Keywords: culverts; culvert hydraulics; skew; misalignment; inlet design; discharge

1. Introduction

Perpendicular crossings between transport infrastructure and natural streams are favored when installing new culverts, but this is not always possible as natural creeks meander through the landscape. Maintaining a perpendicular intersection and not aligning the culvert with the main flow direction leads to reduced flows, higher blockage risks [1–3], and sedimentation and erosion [4–6]. Today, three different methods are used to approach this problem; skewed barrels, upstream realignments, or misaligned structures.

Culvert pipes not perpendicular to the transport infrastructure are called skewed barrels. They allow an alignment of the culvert with the stream, but these designs exceed the minimum length necessary as they do not cross in the shortest possible way. This causes higher costs and often makes different inlet designs necessary as well. In some cases, culvert headwalls are adjusted to the embankment rather than normal to the pipe.

These designs are described as skewed inlets. This method makes the use of pre-cast designs difficult due to the great variety.

The second alternative, a creek or stream realignment, is potentially more expensive than the first solution and requires careful consideration of the surrounding landscape and enough available space.

Directing the stream into the culvert with the help of wingwalls is the third alternative. A commonly used culvert design guideline by Schall et al. [7] does not suggest this solution as they advise against the design of misaligned structures, but the Urban Drainage and Flood Control District (UDFCD) guideline [8] proposes it briefly without detailed instructions. The authors from

the UDFCD [8] recommend adjusting the wingwall angles according to the flow direction. This is supposed to direct the flow into the culvert. A major problem caused by misaligned culverts is the higher risk of sedimentation and erosion around the inlet area. The sudden change of direction causes higher velocities on the outside corner and corresponding lower ones on the inside of the inlet, which leads to erosion of the embankment on one side and sediment buildup on the other [7]. Accumulating sediments restrict the discharge capacities of culverts and it remains questionable if the wingwall adjustment approach from the UDFCD guideline [8] can prevent the issues caused by irregular velocities. In summary, it can be stated that the available information for planners and engineers is very limited when it comes to situations where a misalignment between culvert and stream can not be avoided. The effectiveness of the wingwall method proposed by the UDFCD guideline [8] is unclear, as well as the exact implementation. With basic information missing in this area, the aim of this study is to investigate velocity distributions and turbulence for different misalignment angles and to find ways to optimize the flow path so that negative effects are limited. Computational fluid dynamics (CFD) was used to simulate four different inlet setups at different angles between the stream bed and the culvert. CFD numerically solves the nonlinear partial differential Navier–Stokes equations to model fluid problems. Due to the complexity of the equations, few exact solutions exist today and calculating turbulence correctly is still considered unreasonable [9]. Instead, Reynolds-averaged Navier–Stokes (RANS) equations with turbulence models are used, as this approach allows an approximation to the exact solution averaged over time. The conducted simulations were able to visualize flow paths and boundary layer separation from the culvert walls. Keeping the flow attached to the surrounding structure is an effective way of minimizing headwater loss, as any boundary layer separation causes turbulence and vortex formation downstream [9]. Turbulence is caused by friction and fluid shear and then transferred from large motion scales to continuously smaller ones, eventually dissipated by viscosity [10,11]. This results in lower fluid velocities as the turbulent kinetic energy (TKE, the amount of kinetic energy stored in eddies) is dispersed into heat. The results from the CFD simulations were analyzed for velocity distributions and TKE and compared with a flume setup, where a tracer fluid was used to visualize flow paths. This allowed an investigation of the effectiveness of the proposed solutions over a broad range of misalignment angles.

2. Methods

There is a wide range of possible forms for misaligned culverts with angles varying between 0° to 90° . The natural or a reinforced embankment can end directly next to the culvert inlet or there can be a gap between the opening and the embankment that would be bridged, for example, by a headwall. This research focuses on exploring the flows in a layout where the culvert inlet sits centered in the natural stream bed and the culvert width is a third of the streams width.

2.1. Geometry Setup

The developed setup covers a wide range of the most common possibilities of misaligned culverts while keeping important lengths and ratios of the model the same throughout the different angles, so that the results can be compared between the different setups and misalignments. The four configurations of culverts included: (1) straight headwall, (2) wingwalls adjusted to the flow direction, (3) round crossovers from the inlet to the streambed, and a (4) smooth transition between the opening and the embankment with a spline.

The geometry in Figure 1 (construction described in Table 1) shows the model used in this study. For the CFD simulations, angles from 0° (Figure 1b) to 90° (Figure 1c) were tested with a 10° step size. In addition to these ten angles, a 45° misalignment was tested as well. All these angles were each tested with the following four setups at the inlet, resulting in 44 different CFD simulations.

The standard (*straight*) version was the first configuration tested. It had rectangular corners at the inlet and a headwall perpendicular to the pipe.

The second version tested was named *chamfer*. Two straight lines connected the inlet corners with the sides of the upstream channel, resembling wingwalls. The angle for these wingwalls depended on the degree of misalignment and is described in Table 1, Step 8 and 9. The points P_i created through the intersection between the wingwall and the sidewalls of the channel, were used as reference points for the following two configurations.

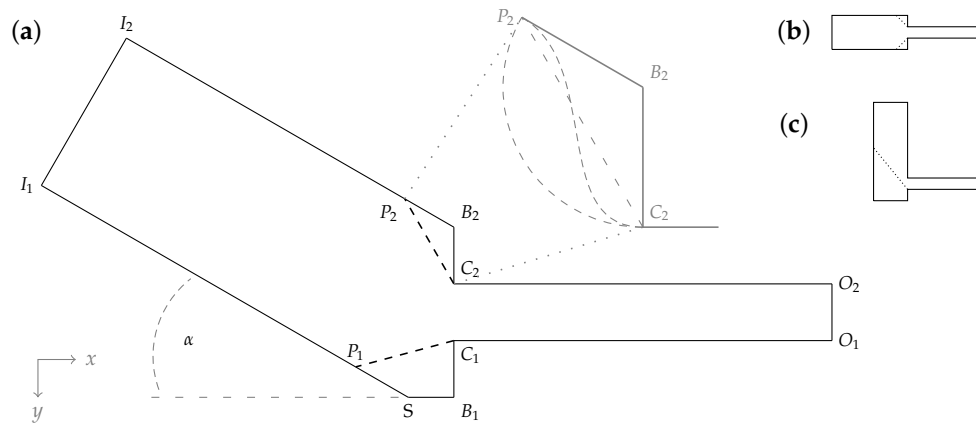


Figure 1. Geometry explanation used for all computational fluid dynamics (CFD) simulations. $C_1C_2O_2O_1$ outlines the culvert, $B_1B_2I_2I_1S$ outlines the upstream water body (a) example geometry with $\alpha = 30^\circ$, (b) no misalignment with $\alpha = 0^\circ$, (c) greatest misalignment with $\alpha = 90^\circ$ (dotted lines in (b,c) indicate the *chamfer*-configuration).

Table 1. Geometry construction for CAD and flume model.

Step	Geometry	Method
1	$\overline{O_1O_2}$	Outlet, $l = 1.5$ m
2	$\overline{C_1O_1}$	Culvert walls, $l = 10$ m, $\perp \overline{O_1O_2}$
3	$\overline{B_1C_1}$	Headwalls, $l = 1.5$ m, $\perp \overline{C_1O_1}$
4	$\overline{B_2I_2}$	Sidewall (headwater body), $l = 10$ m, $\angle C_2B_2I_2 = 90^\circ + \alpha$
5	$\overline{I_1I_2}$	Inlet, $l = 4.5$ m, $\perp \overline{B_2I_2}$
6	$\angle C_1B_1S$	$\perp$
7	$\angle I_2I_1S$	$\perp$
8	$\angle B_1C_1P_1$	$45^\circ + \alpha$
9	$\angle B_2C_2P_2$	$45^\circ - 0.5 \times \alpha$

The construction of the third, *round* version was more complex than the *chamfer* one: Both arcs connected tangential to $\overline{C_1O_1}$ to ensure a smooth transition into the culvert. The arc from C_2 to P_2 was always centered at B_2 , the radius was constant with $r = 1.5$ m. The second arc from C_1 to P_1 continuously changed its radius. The direction of the bend changed from a convex arc for angles smaller than 45° to a concave arc for angles greater than 45° . The radius changed continuously, starting with $r = 1.5$ m at 0° increasing towards 45° and then declining down to $r = 4.5$ m at 90° . As the curve switched its direction at 45° , the radius at this angle is $r = \infty$, a straight line. As a result, the round, 45° setup differs from the other angles as it consists of only one arc instead of two.

The fourth setup (*tangent*) was created with a spline connecting C_i and P_i . Both ends of the spline were set to be tangential to the lines they were connecting to ($\overline{P_iI_i}$ and $\overline{C_iO_i}$), creating a smooth transition between the culvert and the embankment. The spline connecting C_2 and P_2 always changes the sign of the curvature, while the other spline between C_1 and P_1 only changes it up to $\approx 65^\circ$. Greater angles form a convex curve between the two points.

This geometry setup was used for both, the CFD-simulations and the experiments. While the full range of misalignments was modeled in the CFD-analysis, the flume experiments only tested the 45° setup to compare the results with the ones from the simulation.

2.2. Simulation Setup

The CAD software Inventor from Autodesk (Autodesk Inventor Professional 2018) was used to build the computer models, Fluent from ANSYS (ANSYS 16.1) was used to perform the mesh generation, calculate the solution and postprocess the generated data. Table 2 summarizes the configuration details for the numerical simulations. After constructing the two-dimensional models in accordance with Figure 1, they were loaded into ANSYS Meshing. $I_1S B_1 C_1 O_1 / I_1 P_1 C_1 O_1$ and $I_2 B_2 C_2 O_2 / I_2 P_2 C_2 O_2$ were defined as walls. The mesh was inflated from these walls with ten layers, a smooth transition and a growth rate of 1.2. Together with a maximum face size of 60 mm, these settings resulted in 22×10^3 to 29×10^3 hexahedra elements in the different scenarios. The mesh quality was later checked in the solver and improved with a mesh refinement-method until a minimum orthogonal quality >0.75 and a maximum orthogonal skew <0.25 was reached. The orthogonal quality ensures perpendicular vectors within a cell and values close to 1 are considered good. The orthogonal skew serves as a quality indicator for skewness in a mesh, values close to 0 are preferred [12]. Y^+ values were limited to <100 . Minimum values were ignored, as the scalable wall functions ignore $Y^+ < 11$.

Table 2. Simulation parameters for ANSYS Fluent modelling.

Parameter	Configuration
Geometry	Planar 2D
Inlet boundary condition	Velocity inlet, $1 \text{ m}\cdot\text{s}^{-1}$
Outlet boundary condition	Pressure outlet, 0 Pa
Turbulence model	Realizable $k - \epsilon$ with scalable wall functions
Solver	Pressure-based SIMPLE scheme
Time step size	0.1 s

Once the mesh was completed, boundary conditions (inlet velocity = $1 \text{ m}\cdot\text{s}^{-1}$, outlet pressure = 0 Pa) were applied. This corresponded to a test condition flow rate of $5 \text{ L}\cdot\text{s}^{-1}$. The wall material was defined as alloy to minimize turbulence creation from friction. For the turbulence modeling, the realizable $k - \epsilon$ model was chosen with scalable wall functions. The basic $k - \epsilon$ -model calculates the turbulence-viscosity transport with two equations, one for the turbulent kinetic energy k and one for the isotropic dissipation rate ϵ . The further developed realizable $k - \epsilon$ -model was chosen for the conducted simulations, as it offers higher accuracy in jet prediction and supposedly performs better in flows involving separation, recirculation, and boundary layers under strong adverse pressure gradients [12,13]. This is achieved through a new equation replacing the constant C_μ and a refined definition of the dissipation rate ϵ [13].

Another simplification in many CFD models is the assumption of isotropic turbulence. There is a notable reduction in complexity with little loss of accuracy, but as this assumption leads to incorrect results in near wall regions, Launder and Spalding [14] introduced a set of wall functions to compensate for these limitations. The scalable wall functions used in this study require a mesh where $y^* \leq 100$, so that the viscous sublayer and the buffer layer can be modeled correctly according to the law of the wall [15].

All simulations were solved in a planar 2D space, pressure based and with an absolute velocity formulation. Transient calculations with a time step size of 0.1 s were performed until a stable result was reached for TKE. To control this, the sum of TKE over the fluid body was calculated and monitored over a minimum of 1000 timesteps for all different 90° misalignments. The 90° setups were chosen for this observation as it was assumed that they would need the most time to converge. Figure A1 shows the course of these sums. Following from this, all remaining chamfer, round and tangent scenarios ran

for a total of 90 s while the straight scenarios ran for 150 s. The change in total TKE from 150 s to 300 s was less than 0.5%. It was therefore decided that sufficient accuracy was achieved after 2.5 min.

The simulation results were examined for three different characteristics, (1) interfering flows inside the culvert, (2) velocity distributions, and (3) TKE. Interfering flows inside the culvert are the ones flowing in other directions than the main one. This causes turbulence downstream and energy loss. To analyze these flows, velocity vectors were dissected into their x- and y- components so that the parts interfering with the main flow ($\vec{y}$) were isolated.

The velocity distributions were used to analyze how much available culvert width was utilized for discharge. The uniform distribution throughout the barrel allows a greater overall flow rate and reduces problems associated with erosion and sedimentation. The overall TKE in any model was used as a third performance indicator. The absolute values allowed a direct comparison between different scenarios and lower TKE values indicated a greater discharge capacity.

These three analyses were employed to reveal the advantages and disadvantages of the proposed misalignment treatments. To verify the results of the conducted simulations, the 45° misalignment studies were compared to flows in an experimental setup.

2.3. Experimental Setup

A CNC-router was used to build perspex models based on the numerical simulations with 45° misalignment. These models were placed in a 2.5 m wide and 8 m long channel (Figure 2). Water was stored in an underground tank and pumped from there into the channel. The inflow was designed as a weir overflow across the whole channel width and the water was permitted to discharge freely off the other end. The flow rate was controlled via a variable speed drive and measured with a magnetic flow meter (WaterMaster - FEX100, ABB Australia, Brisbane). For all experiments presented in this paper, the flow rate was set to $5 \text{ L} \cdot \text{s}^{-1}$. Diluted rhodamine was used as tracer, which was dropped into the flowing water with a multichannel pipette along $\overline{I_1 I_2}$. The created streak lines were used to compare the experiments with the simulations.

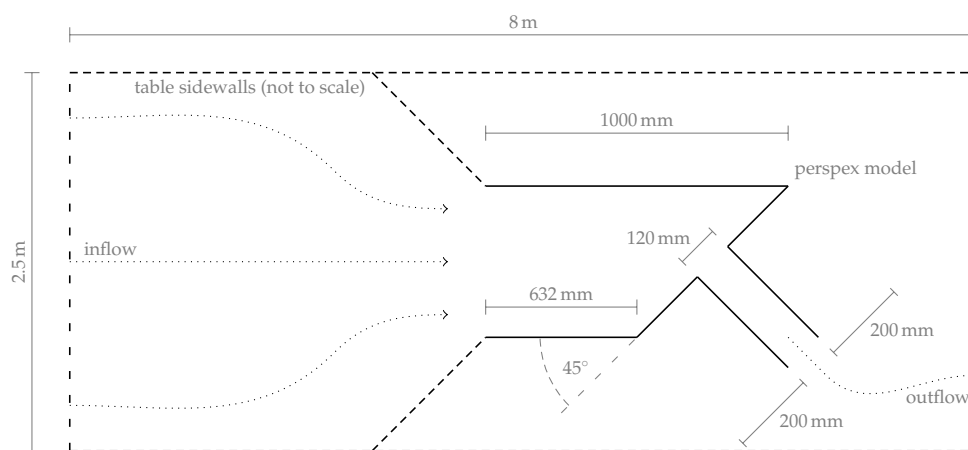


Figure 2. Experimental layout (not to scale).

3. Results and Discussion

The following section summarizes the results and findings from the simulations and experiments. Figure 3 contains the results of all simulations, and only limited CFD-visualizations are contained in this section (Figure 4). A detailed flow presentation of all simulated misalignment angles can be found in the Appendix A, Figures A2–A5.

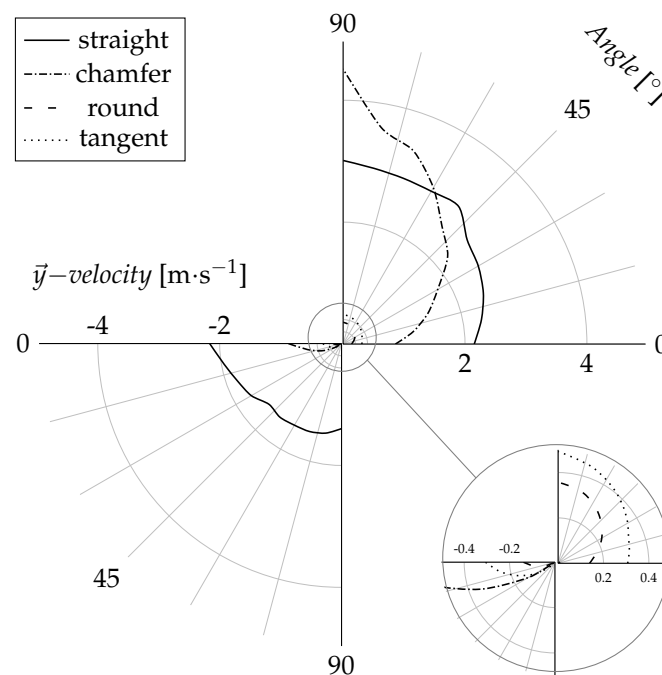


Figure 3. Maximum and minimum velocities in $\vec{y}$ -direction sampled from $\overline{C_1C_2}$ for all culvert angles.

3.1. Simulation Results

Figure 3 shows velocities in the y -direction sampled along $\overline{C_1C_2}$. These velocities perpendicular to the main flow direction ($\vec{x}$) cause turbulence downstream. In an aligned setup (0°), the maximum and minimum velocities have the same magnitude but different signs. The wingwalls (chamfer-option) are able to reduce the amount and size of those shear velocities by more than 50% compared to a setup with a straight headwall. But this changes for misaligned structures. While the maximum $\vec{y}$ -velocities from the straight setup only increase by about 50%, they quintuple in the chamfer setup. Velocities in $\vec{y}$ -direction continuously increase in the chamfer setup and beyond 60° those perpendicular velocities exceed the ones from the straight setup.

The *round* configuration maintains the lowest shear velocities out of all cases and for every angle. It maintains smaller (between 25%–60%) velocities than the *tangent* option, but both of these configurations have significantly lower $\vec{y}$ -velocities than the *straight* and *chamfer* ones.

The second part in analyzing the modeling results was a visualization of the velocity distributions in the culvert. Figure 4 uses velocity contours to visualize the impact of different inlet treatments. Three different cases are presented, 0°, 45° and 90°, and the velocities are differentiated with different shades of blue. Figure 4a has a higher velocity jet in the culvert than Figure 4b–d as there is a sudden change in width that causes a constriction downstream. The *chamfer* setup creates a homogenous velocity distribution in an aligned culvert but it fails to improve the flow within the culvert when the misalignment angles increase (Figure 4f,j). Here, the lower wingwall constricts the flow in the upstream water body more and more, leading to a bottleneck that is narrower than the actual culvert. All *round* and *tangent* setups (Figure 4c,d,g,h,k,l) form smooth velocity fields in the culvert. Even at large angles they only develop small, similar sized areas of high velocities along the upper inlet (Figure 4k,l).

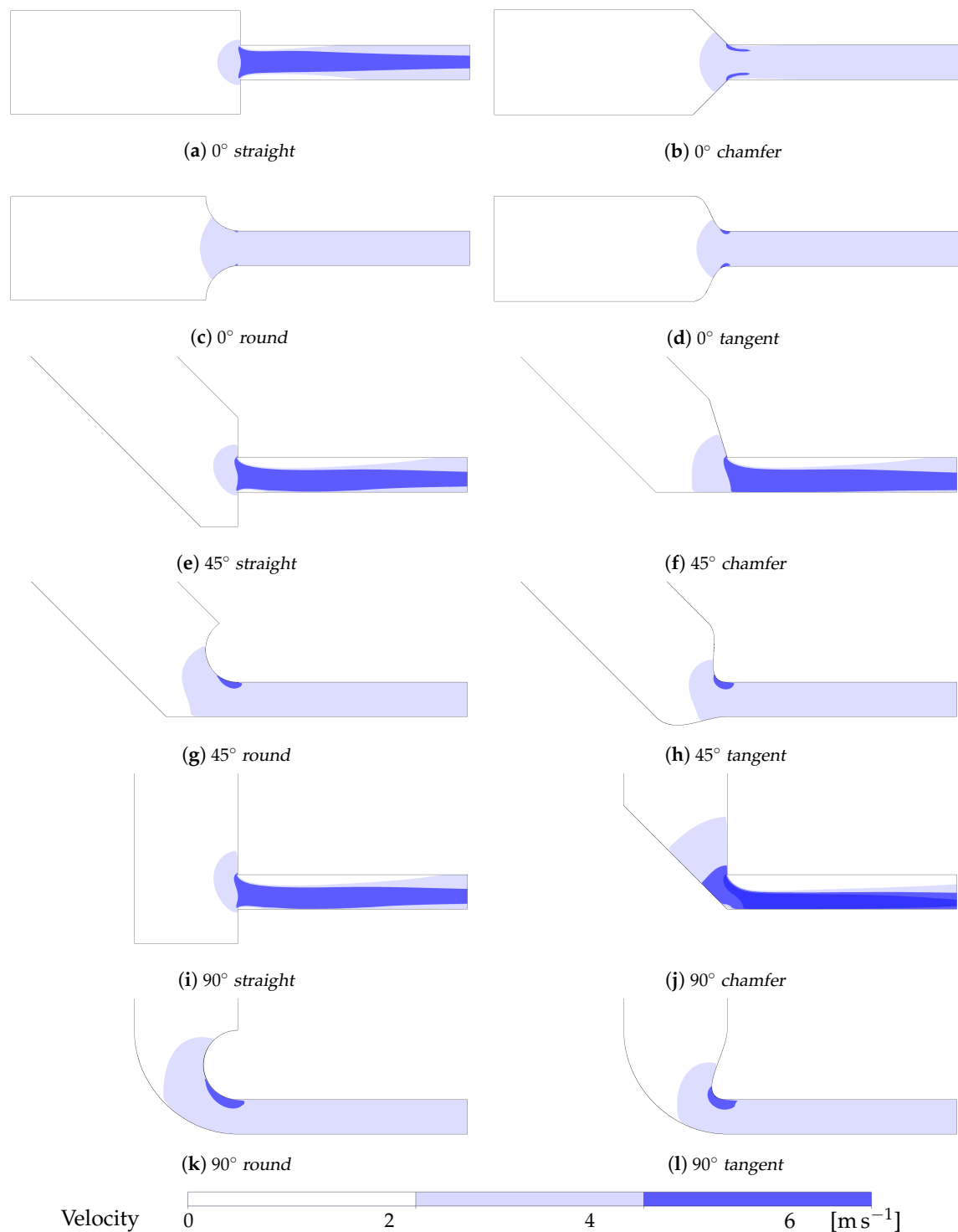


Figure 4. Comparison of different velocity contours.

Figure 5 shows the sum of TKE over the total area in all tested scenarios. Although the total area of the model varies with every case, its influence on the sum of TKE is rather small. Most of the TKE occurs in the culverts, which all had the same size. There is a linear trend for the *straight* configuration, with $R^2 = 0.964$. Out of all four configurations, this one has the second highest rise in TKE with increasing misalignment: from 0° to 90° TKE increases by 22.75% ($236 \text{ m} \cdot \text{s}^{-2}$). For aligned culverts, the *chamfers* or wingwalls reduce turbulence by about a factor of five in comparison to the

straight setup, however, the proposed scheme does not perform well for other angles, especially steeper ones. The TKE increases exponentially ($R^2 = 0.995$) more than ten times from aligned to perpendicular flows. This is due to the fact that the bottom wingwall narrows down the flowpath in the upstream water body at greater angles, therefore creating a bottleneck. In its most extreme case, the 90° scenario, this leads to a high velocity jet formation before the culvert with velocity vectors at least 45° off the culvert flow direction. This leads to a highly irregular flow pattern in the culvert with a large amount of turbulence. Different wingwall schemes that do not create such a constriction would most likely perform better. Extending the lower culvert sidewall and thereby dissolving the bottleneck could be one solution, trialling different angles another one. It is questionable whether a straight wingwall setup could perform continuously as well as the *round* or *tangent* version. Both of these latter setups start off with low losses and maintain these as the angles get bigger. Between 0° to 90° TKE increases just by 20% from approximately 100 m to $120 \text{ m} \cdot \text{s}^{-2}$ in the *round* and *tangent* setup (quadratic increase; round: $R^2 = 0.992$; tangent: $R^2 = 0.974$). The greatest deviation between the round and tangent version is 3.1%.

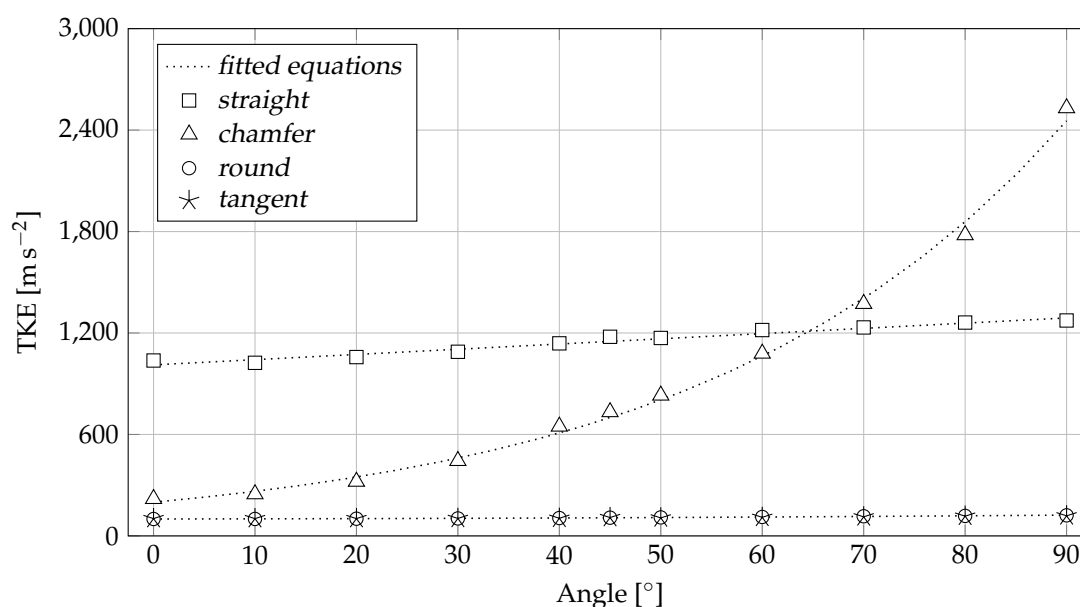


Figure 5. TKE sums for every case. All relations between energy loss and angle of alignment were significant ($p < 0.001$).

Until today, guidelines advise against any misalignment between the stream and the culvert and there is little information on what to do when a misalignment cannot be avoided. The Urban Drainage and Flood Control District in the US recommends the use of wingwalls in cases the culvert is skewed to the normal channel flow [8]. Our experiments showed that rounded or tangent transitions would in all cases exceed the performance of the suggested wingwall modifications. The round and tangent modifications reduced turbulence during the flow direction change and a more even velocity distribution was achieved. This reduces sedimentation and erosion problems and is therefore a preferable solution over the wingwalls.

3.2. Experimental Results

Streaklines in the four different setups were visualized with diluted Rhodamine as a tracer. This helped to identify smooth flows as well as areas of very high and low flow velocities.

In the straight configuration (Figure 6 s_l and s_r) we were able to monitor a long residue time in both corners next to the inlet. This longer retention time was also seen in the left corners of the chamfer and tangent setup (Figure 6 c_l and t_l). The round setup (Figure 6 r_l and r_r) did not accumulate tracer at any point and had the smoothest flowpath out of all the setups. The round setup offers a steady

change in direction combined with a consistent narrowing of the flow path. This is presumed to be the reason for its superior performance over the other setups. While the tangent version had no sudden changes in direction either, the upstream channel width changes much quicker than in the round setup. This creates the mentioned low velocity area in front of the culvert where sediments can accumulate.

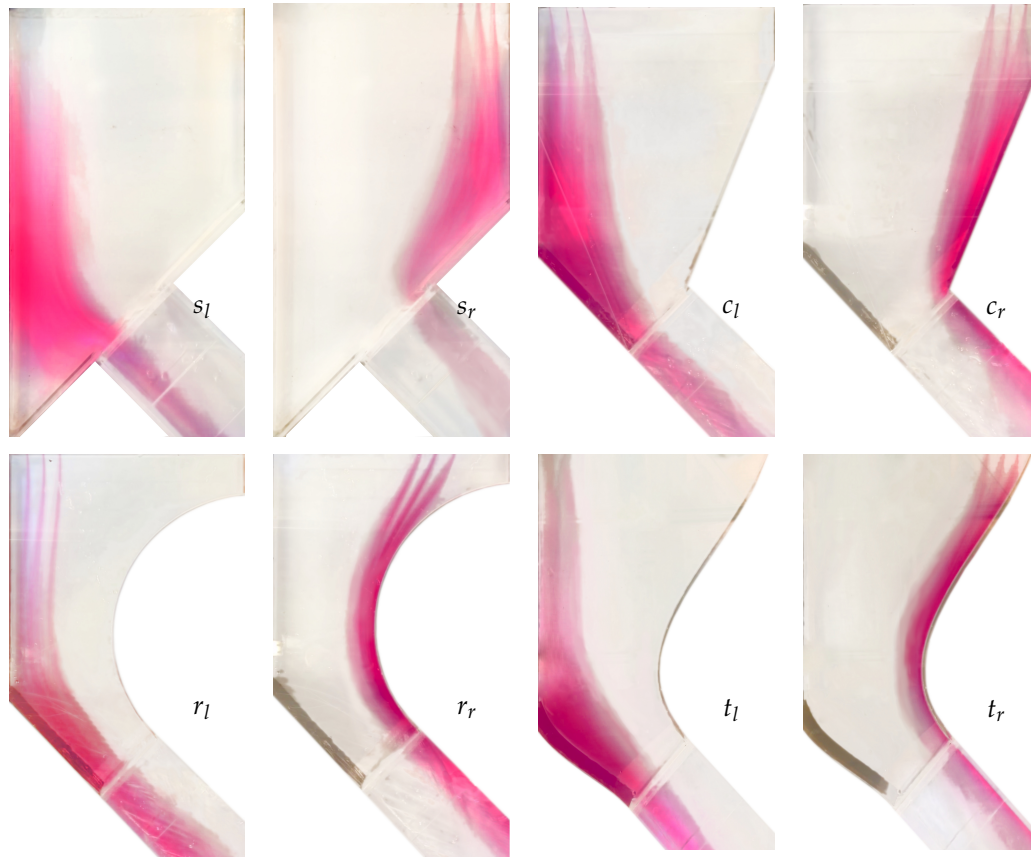


Figure 6. Results from the inlet experiments; pictures show a long exposure of the tracer at $5 \text{ L} \cdot \text{s}^{-2}$ ($t = 3 \text{ s}$); s = straight, c = chamfer, r = round, t = tangent; l = left, r = right.

Headwater heights (h) were measured upstream of the inlet with a thin metal ruler. The water levels in the round ($h = 85 \text{ mm}$) and tangent ($h = 87 \text{ mm}$) setup were lower than the chamfer ($h = 90 \text{ mm}$) and straight ($h = 93 \text{ mm}$) one, supporting the simulation results.

Presenting these results in terms of dimensionless depth, h/D and dimensionless flow rate Q^* permits a comparison of these scaled results to full-scale culvert experiments through Froude similitude [16]. Dimensionless flow rate is defined as $Q^* = Q / \sqrt{(gD^5)}$ where g is the acceleration due to gravity. The dimensionless water depths for the four cases were 0.71 (round), 0.73 (tangent), 0.75 (chamfer), and 0.78 (straight). Dimensionless flow rate was 0.32 for all experimental results. The experimental results were used as a tool for qualitative validation of the numerical simulations. In general, the experimental results agreed with the trends found in the simulations, showing relative improvements in the smooth transition cases (round and tangent) when compared to the sharper geometries (tangent and straight). In modern culvert design there is often a gap between the wingwalls and the inlet. This could have been another alternative in the simulations, but was not chosen as it is known to be a less effective training method than a flush wingwall connection to the inlet [7]. Different sized inlet configurations will perform differently as well, but a size comparison was beyond the scope of this study.

4. Conclusions

The simulations and experiments have shown that misaligned culverts can maintain a uniform velocity distribution and high discharge capacity when flow is smoothly directed into the structure. Rounded and tangential transitions between the upper water body and the culvert had the biggest positive impact on the flows for any degree of misalignment. These methods increase the flow capacity while also decreasing the likelihood of blockage and reducing sedimentation and erosion processes due to less vorticity upstream and a more uniform velocity distribution [2].

This has potential benefits for future culvert constructions, as well as existing structures. Cost savings are possible as shorter barrels perpendicular to the embankment can be used and creek realignments could potentially become unnecessary. Smooth transitions of flow directions work better than sharp bends or corners and a steady reduction of the channel width down to the culvert width will ensure a uniform flow pattern. The chamfer configuration resembling wing walls improves the flow while the alignment deviations are small, but the method fails for greater misalignments. The authors advise against this method as the presented round and tangent alternatives continuously perform better.

This research did not examine three-dimensional structures or transitions and more information is needed in this area. As all sudden changes in direction should be avoided, smooth transitions from a u-shaped channel to a rectangular box culvert or a round pipe are necessary. Further research in this area should ensure even velocity distributions as far as possible so as to avoid the formation of erosion or sedimentation zones. Last but not least, it should be mentioned that the influence of the top edge on the discharge capacity is substantial [17,18]. Therefore, the top bevel should be incorporated in any attempt to increase the flow capacity to ensure sufficient discharge in unsubmerged and submerged conditions.

Author Contributions: Conceptualization, formal analysis, investigation, methodology, software, validation, Writing—Original draft preparation, R.J.; Writing—Review and editing, R.J., C.J., K.T. and N.T.; supervision, C.J., K.T. and N.T.

Funding: A scholarship and operational funds were provided by the Sunshine Coast Council, and we very much appreciate their support.

Acknowledgments: The authors thank Madelyn Kilby for her work.

Conflicts of Interest: The authors declare no conflict of interest. The funders had no role in the design of the study; in the collection, analyses, or interpretation of data; in the writing of the manuscript, or in the decision to publish the results.

Appendix A

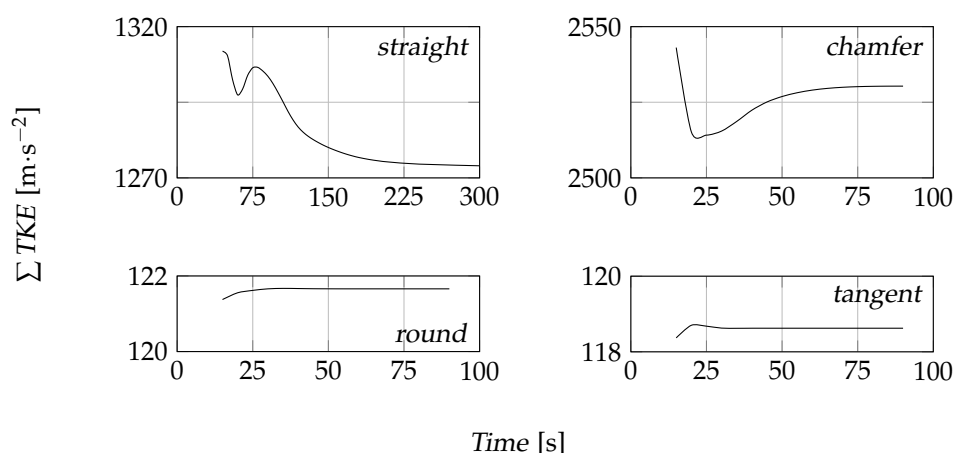


Figure A1. Convergence plot for Turbulence Kinetic Energy (TKE) in transient simulations with time step size 0.1 s and 90° misalignment. These plots were used to determine the minimum time required for the transient simulations.

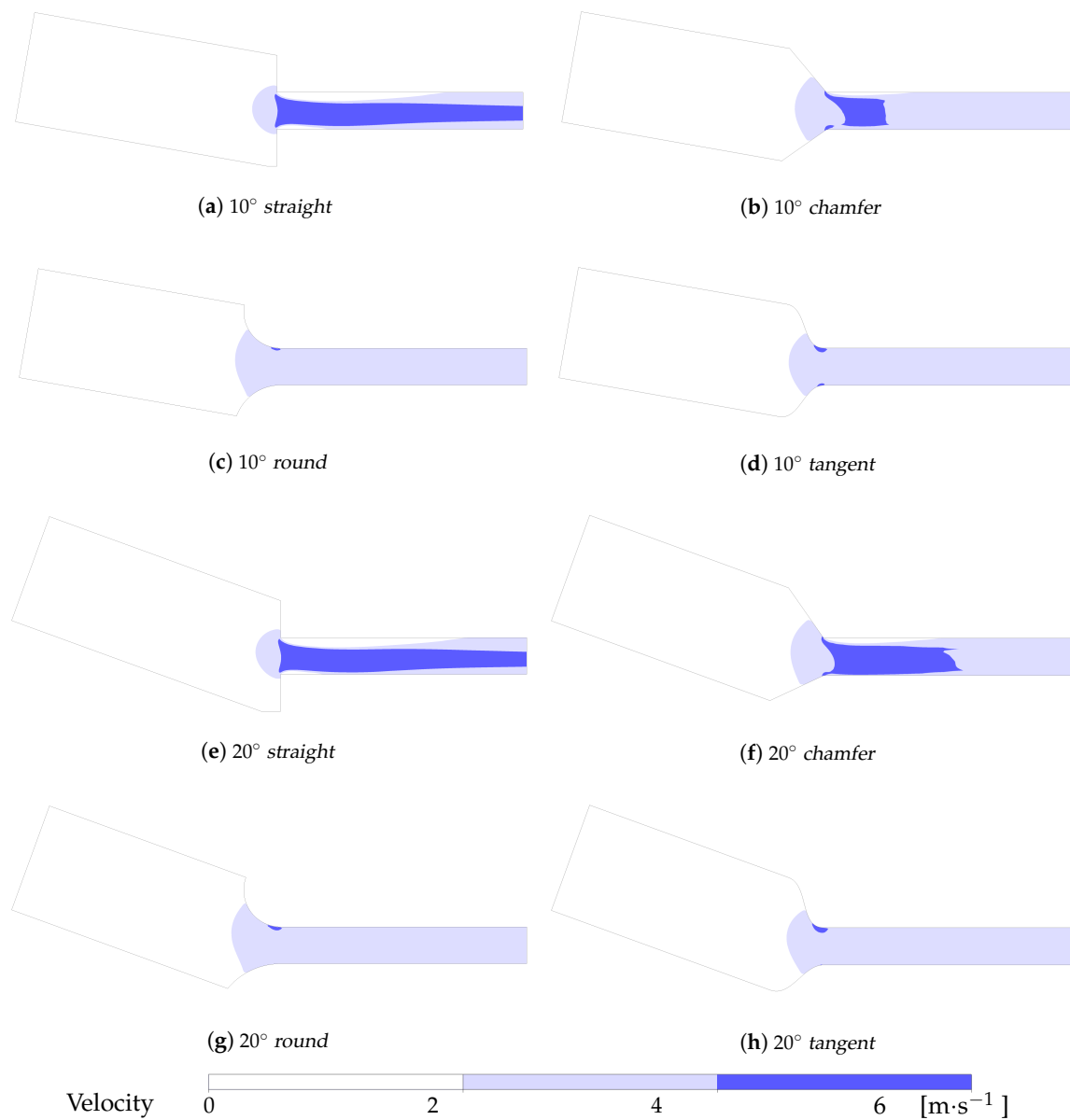


Figure A2. Velocity Contours for 10° and 20° misalignments.



Figure A3. Velocity Contours for 30° and 40° misalignments.

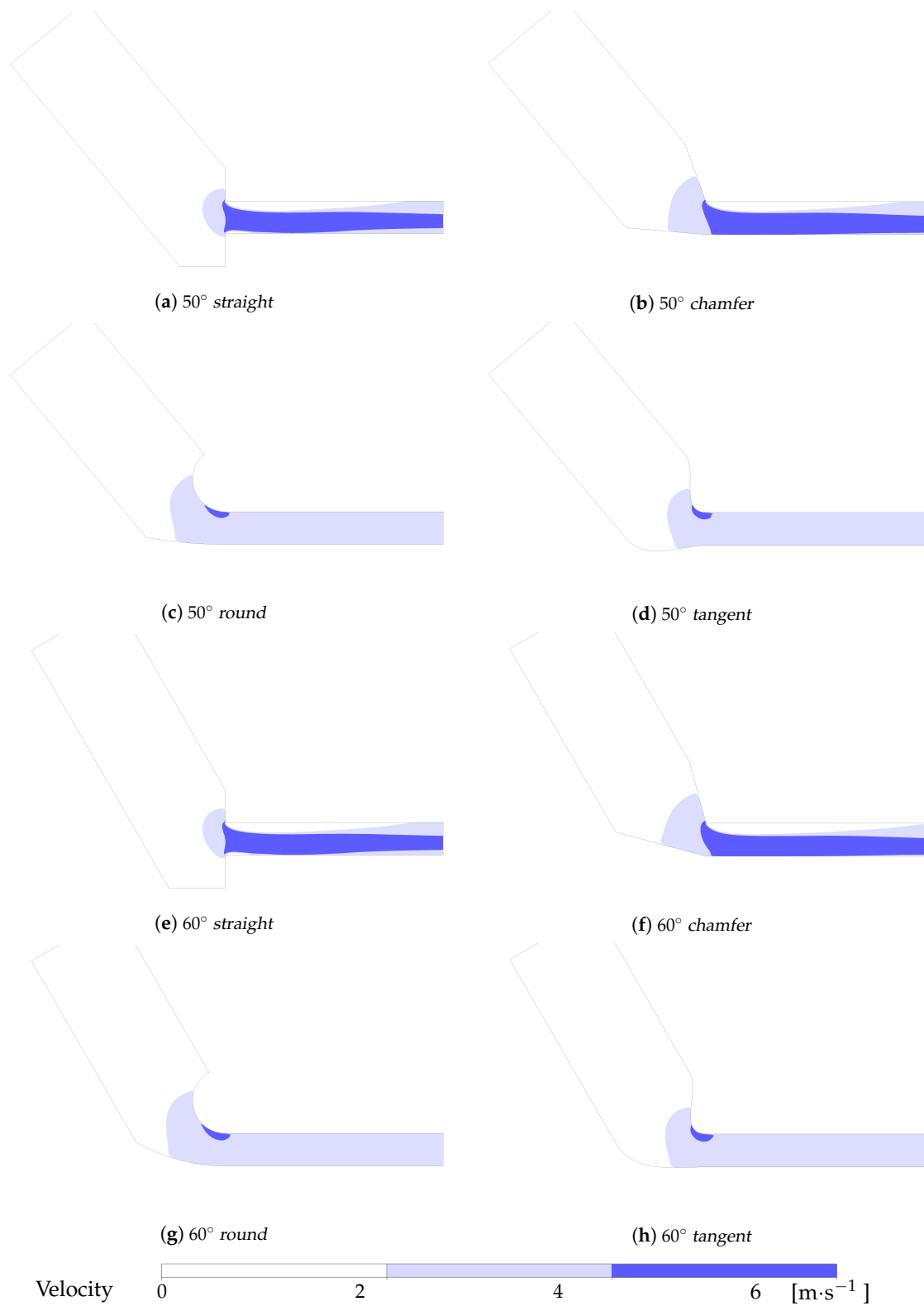


Figure A4. Velocity Contours for 50° and 60° misalignments.

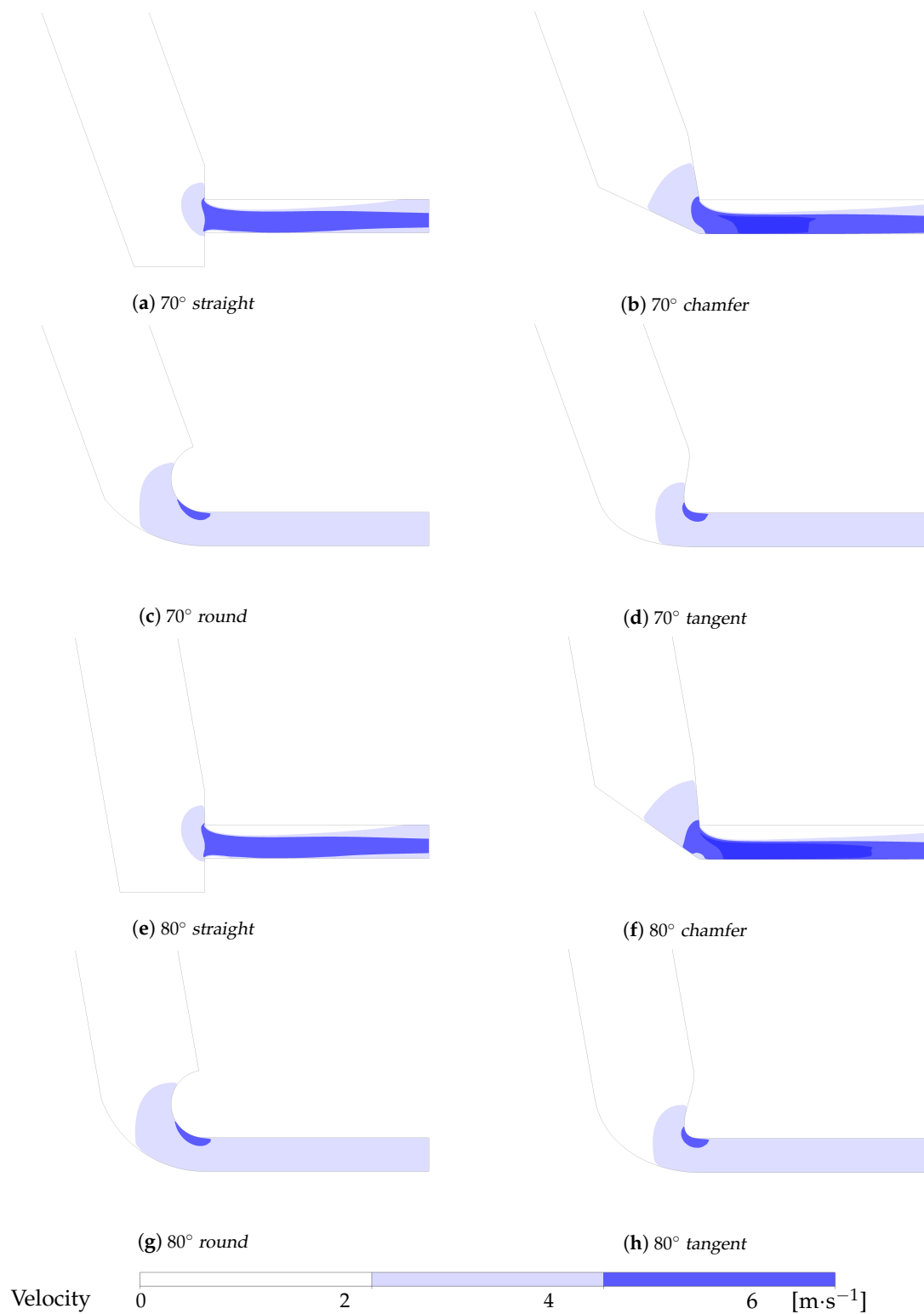


Figure A5. Velocity Contours for 70° and 80° misalignments.

References

1. Allen, D.; Arthur, S.; Haynes, H.; Wallis, S.G.; Wallerstein, N. Influences and drivers of woody debris movement in urban watercourses. *Sci. China Technol. Sci.* **2014**, *57*, 1512–1521. [[CrossRef](#)]
2. Kang, N.; Kim, S.; Kim, Y.; Noh, H.; Hong, S.; Kim, H. Urban Drainage System Improvement for Climate Change Adaptation. *Water* **2016**, *8*, 268. [[CrossRef](#)]
3. French, R.; Jones, M. Design for culvert blockage: The ARR 2016 guidelines. *Aust. J. Water Resour.* **2018**, *22*, 84–87. [[CrossRef](#)]
4. Kolerski, T.; Wielgat, P. Velocity field characteristics at the inlet to a pipe culvert. *Arch. Hydroeng. Environ. Mech.* **2014**, *61*, 127–140. [[CrossRef](#)]
5. Rowley, K.J.; Hotchkiss, R.H. Sediment Transport Conditions Near Culverts. Master's Thesis, Brigham Young University, Provo, UT, USA, 2014.
6. Kim, Y.; Paik, J. Depositional characteristics of debris flows in a rectangular channel with an abrupt change in slope. *J. Hydro Environ. Res.* **2015**, *9*, 420–428. [[CrossRef](#)]
7. Schall, J.D.; Thompson, P.L.; Zerges, S.M.; Kilgore, R.T.; Morris, J.L. *Hydraulic Design of Highway Culverts*, 3rd ed.; Hydraulic design series; U.S. Dept. of Transportation, Federal Highway Administration: Washington, DC, USA, 2012; Volume 5.
8. UDFCD. *Urban Storm Drainage Criteria Manual Volume 2: Structures, Storage and Recreation*; Urban Storm Drainage Criteria Manual; Urban Drainage and Flood Control District: Denver, CO, USA, 2016.
9. Spalart, P. Strategies for turbulence modelling and simulations. *Int. J. Heat Fluid Flow* **2000**, *21*, 252–263. [[CrossRef](#)]
10. Frisch, U.; Kolmogorov, A.N. *Turbulence: The Legacy of A.N. Kolmogorov*/Uriel Frisch; Cambridge University Press: Cambridge, UK, 1995.
11. Richardson, L.F. *Weather Prediction by Numerical Process*; Harpress Publishing: Los Angeles, CA, USA, 2012.
12. ANSYS Fluent. *Theory Guide*; Ansys Inc.: Canonsburg, PA, USA, 2015.
13. Shih, T.H.; Liou, W.W.; Shabbir, A.; Yang, Z.; Zhu, J. A new k-epsilon eddy viscosity model for high reynolds number turbulent flows. *Comput. Fluids* **1995**, *24*, 227–238. [[CrossRef](#)]
14. Launder, B.E.; Spalding, D.B. The numerical computation of turbulent flows. *Comput. Methods Appl. Mech. Eng.* **1974**, *3*, 269–289. [[CrossRef](#)]
15. von Kármán, T. Mechanische Ähnlichkeit Und Turbulenz. *Sonderdrucke Aus Den Nachrichten von der Gesellschaft der Wissenschaften zu Göttingen: Mathematisch-Physische Klasse; Fachgruppe 1, Nr 5 u. Fachgruppe 2, Nr 1*; Weidmannsche Buchh: Berlin, Germany, 1930; Volume 1930.
16. Hager, W.H.; Del Giudice, G. Generalized culvert design diagram. *J. Irrig. Drain. Eng.* **1998**, *124*, 271–274. [[CrossRef](#)]
17. Harrison, L.J. *Hydraulic Design of Improved Inlets for Culverts*; United States Federal Highway Administration Hydraulic Engineering Circular; U.S. Federal Highway Administration: Washington, DC, USA, 1972; Volume 13.
18. Jones, J.S.; Kerenyi, K.; Stein, S. *Effects of Inlet Geometry on Hydraulic Performance of Box Culverts: Laboratory Report*; U.S. Dept. of Transportation, Federal Highway Administration, Turner-Fairbank Highway Research Center: McLean, VA, USA, 2004.



© 2019 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<http://creativecommons.org/licenses/by/4.0/>).