CFD MODELLING OF STORMWATER OVERLAND FLOW AT THE ENTRANCE TO A ROAD UNDERPASS

K G Heays[1,2], S Reed[1,3], T S R Fisher[1,3]
[1]Memorial Park Alliance
[2]Tonkin & Taylor Ltd, 265 Wakefield St, Wellington
[3]Tonkin & Taylor, 105 Carlton Gore Road, Newmarket, Auckland

ABSTRACT

The Buckle Street underpass is part of the Memorial Park project being delivered by the Memorial Park Alliance. Investigations of stormwater systems for the proposed Buckle Street underpass indicated that overland flows posed a flood risk to the underpass. Star CCM+ Computational Fluid Dynamics (CFD) software was used to model the stormwater flow through the intersection. The flow of water entering and bypassing the underpass was quantified.

The site is located at the intersection of Taranaki, Buckle and Arthur Streets, Wellington. The Taranaki Street catchment above the intersection is 23.3 ha and relatively steep with a high proportion of impervious surfaces. The overland flows entering the intersection were predicted to be 5.4 m^3/s. Some of the overland flow from Taranaki Street will be diverted down the underpass. The design of the intersection and underpass drainage has been designed to accommodate these overland flows.

CFD modelling was undertaken to refine the intersection surface design, to assist in striking the best balance between a safe road alignment and flood protection. The exact nature of CFD modelling allows the evaluation of the effect of small changes in camber on the volume of water entering the underpass. The results from this study show an encouraging progression in the application of CFD to complex overland flow applications in engineering.

KEYWORDS

CFD, Modelling, Roading, Stormwater

1 INTRODUCTION

The Buckle Street underpass is part of the Memorial Park project being delivered by the Memorial Park Alliance. The project is to underground the Buckle Street section of SH1 in Wellington. The project is to remove the traffic that currently separates Memorial Park from the National War Memorial, and create a new, unified National Memorial precinct. This project is a key part of the Ministry of Culture and Heritage’s commemoration of the centenary of the First World War, and the completed New Zealand Memorial Park will be in place by Anzac Day 2015. The project is being delivered to NZTA by the Memorial Park Alliance.

The site is located at the intersection of Taranaki, Buckle and Arthur Streets, Wellington. The Taranaki Street catchment above the intersection is 23.3 ha and relatively steep with a high proportion of impervious surfaces. The flows entering the intersection were predicted to be 6.4 m^3/s in the 500 year event (based on guidelines from WCC 2012). The maximum network capacity was conservatively estimated to be 1 m^3/s, resulting in an overland flow volume of 5.4 m^3/s to enter the intersection in the 500 year event. Flows for the 100 yr and 10 year event were also investigated. The proposed Buckle Street underpass has approaches that start at the intersection.

The overland flows at the intersection were identified as a high project risk due to the potential for flooding of the underpass. The design of the intersection and underpass drainage needs to accommodate these overland flows. The objective for the intersection surface design was to strike the best balance between a safe road alignment and flood protection.

Uncertainty around the flow of water entering the underpass was generated, as the overland flow spilt into the underpass is dependent on both the water momentum and the geometry of the intersection. The high flows down Taranaki Street, sloping roads and cross cambers, inflow from a minor road and the intersection geometry are major factors in determining the flow that enters the underpass.

Evaluation of the flows through the intersection could not be done using traditional analytical models. Weir assumptions are not appropriate because the hydraulics involve the bifurcation of super-critical, shallow flows. The bifurcation of flow into Buckle Street occurs at a right angle. Small changes in the road surface affect the volume of water entering the underpass. The spatially varying nature of this problem requires modelling in at least two dimensions.

CFD models refer to the use of numerical models to solve the Navier–Stokes equations using higher order numerical models with relatively few simplifications. A range of CFD models are available, which have a range in complexity and corresponding level of approximations made to solve fluid mechanics equations. The most advanced models generally contain multi phase fluid, take into account thermodynamic effects and are particularly well suited to determining flow behaviour in turbulent conditions (Toombes & Chanson 2011).

CFD is a widely accepted numerical tool that enables the accurate predictions of fluid dynamics (Vega et al., 2003, Shilton et al., 2008). CFD has been successfully verified against physical models in other stormwater applications, such as the fluid mechanics of stormwater detention ponds (Khan et al., 2013) and in wider engineering applications such as chemical engineering (Vakili & Esfahany 2009, Wu 2011) and mechanical engineering (Gräf & Kleiser 2011; Jain et al. 2011). The advantage of CFD is that it enables fast visualisation of complex flow features, while allowing a large range of geometric and flow scenarios to be readily considered.

All CFD solutions were calculated in STAR-CCM+ (v7 or higher), a momentum coupled finite volume CFD code used extensively in process, plant, power, automotive, aerospace and environmental industries. CFD solves Reynolds Averaged Navier Stokes (RANS) equations to predict the movement of fluids. For modelling the free surface, StarCCM+ uses the Volume of Fluid (VOF) method. This feature allows the program to track the free surface of the water, allowing a realistic interaction between the water and air phases.

2 MODEL DEVELOPMENT

2.1 SITE LAYOUT

The modelled area lies at the intersection of Taranaki Street, Arthur Street and Buckle Street in Wellington, NZ. Taranaki Street runs in a north-east direction and enters the intersection at an approximate slope of 1%. Arthur Street slopes down toward Taranaki Street from the west, and the proposed underpass will be constructed along Buckle Street to the east, perpendicular to Taranaki Street. The catchment draining to the intersection includes primarily urban areas in the lower catchment, and steeper vegetated slopes that are part of the town belt in the upper catchment.

The design layout of the intersection includes a crossfall of 0.5% along Taranaki St away from the Buckle St underpass entrance to minimise the flow into the underpass. Other drainage and geometric interventions to minimise the flow into the underpass were considered, but were not found to be viable or satisfactory.

Factors affecting the choice of model extent were:

- Extent to which the new intersection alignment was created
- Minimise model extent to reduce model run times
- Rotation of the grid to reduce overall model extent and provide an alignment with the inlet boundary conditions

Figure 1 shows the data used to create the model geometry, with the 500 yr peak overland flows estimated to enter the intersection.

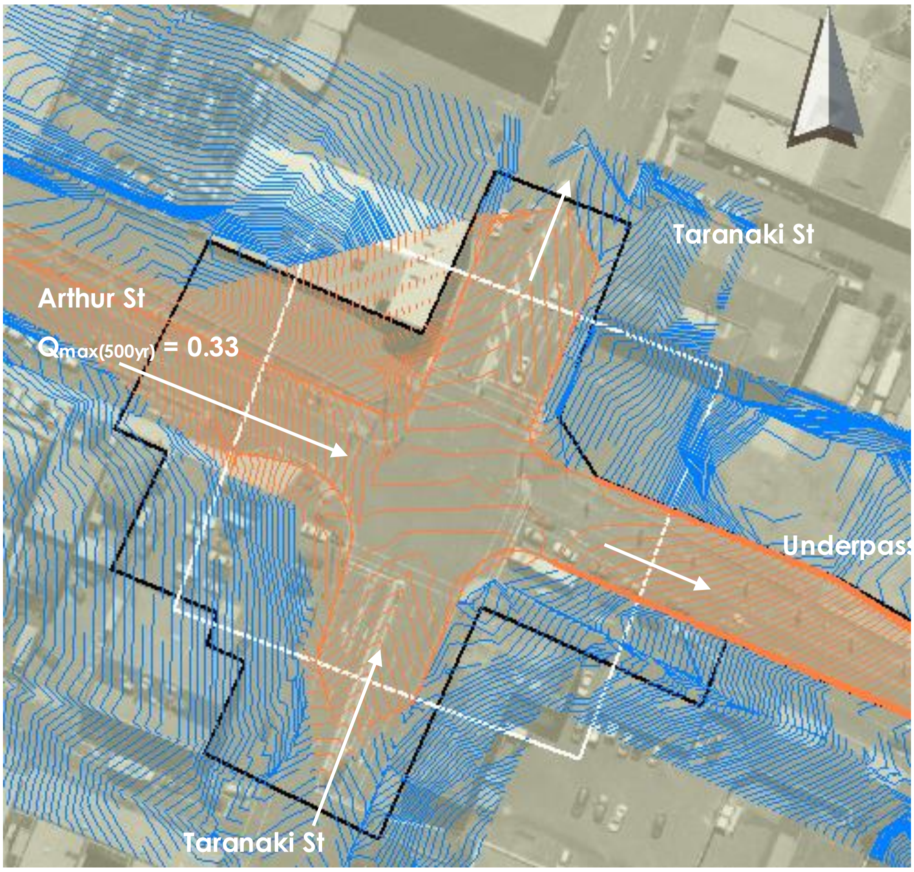

Figure 1. Site layout and source data, where the orange contours show the 0.05 m contours of the new road alignment, blue contours show the 0.05 m contours of the existing land surrounding the intersection, black shows the area of interest and white shows the final model extent which was chosen to minimize model surface area.

2.2 MODEL GEOMETRY

The surface input to StarCCM+ was generated using contour and .TIN files in the program Global Mapper. A combination of the design intersection alignment and existing contours were used to create the modelled surface. Smooth surfaces are important for the model, so some smoothing of the land surrounding the intersection was done. No adjustment was made to the design intersection road surface or kerbs.

The underpass walls created an unacceptable level of roughness, therefore artificial smooth walls were generated in their place. These were of the same shape and in the same position as the designed walls. Figure 2 shows the artificial walls. The model surface was rotated 20^{o} so the model aligned in the xy direction

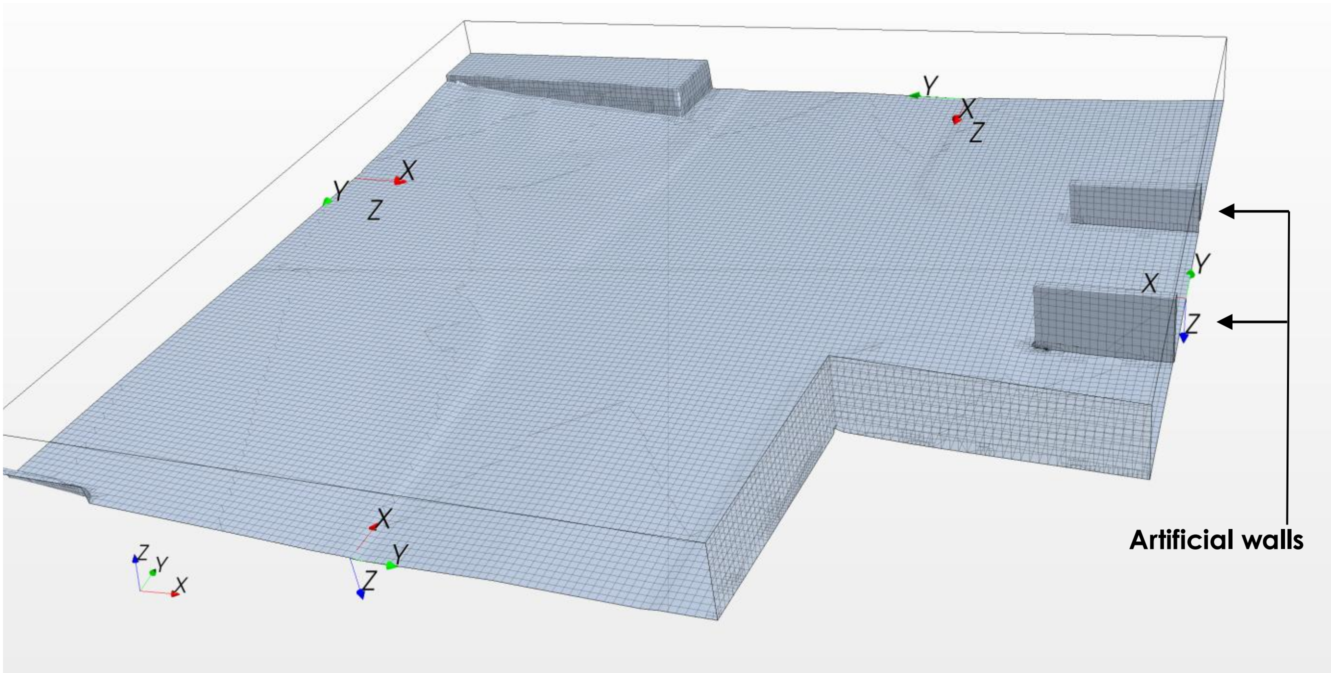

Figure 2. Model mesh: The walls and road surface as represented in the model. These surfaces represent the bottom and side of the mode that contain the simulated flow. Artificial walls at the tunnel entrance were used to overcome unacceptably high roughness from the design data at these locations.

The model geometry was created inside StarCCM+. The imported surface was used as a base, on top of which a volume was generated to create the model domain. Numerical modelling requires that the model domain be discretised into cells to allow for numerical solution to be calculated for the model domain (Figure 2).

The 3D model was created using a Trimmed Hexahedral Mesh with second spatial discretisation (Figure 3). This mesh is designed to:

- Create predominantly hexahedral mesh with minimal skewness
- Allow curvature and refinement wherever necessary at the model surfaces

Reduction of the mesh size allows for greater definition of the model surface and increases the number of calculations within the model, therefore increasing accuracy in the result. A larger grid size reduces the model run times by reducing the number of calculations required to reach a solution. In order to build a realistic model that was completed in a timely manner, using a reasonable time step, a compromise between accuracy and model run times was required.

Modelling of the flow also requires refinement of the grid in the boundary layer of the water near the ground surface (Figure 4). The refinement in this region is called 'prism layer meshing'. The prism layer meshing is applied to all solid boundaries, i.e. the floor and walls of the model. The prism layer thickness is the thickness in the vertical direction that is subject to the prism layer meshing refinement. This value should be large enough to comfortably include the boundary layer. The prism layer number is the number of divisions that the prism layer is separated into (Figure 4). The prism layer stretching value represents the rate of increase in size as cells move away from the boundary layer. For example, a prism layer stretching value of 1.1 means that each layer is 1.1 times the thickness of its outside neighbour layer. The size of the prism layer mesh affects the validity of the RANS calculations, therefore a sensitivity analysis on a test run was conducted to determine appropriate values. See table 1 for a summary of the mesh sizes.

Table 1. Mesh size

Variable	Units	Value
Mesh size	m (x,y,z)	0.5
Prism layer thickness	m (z)	0.4
Prism layer number		40
Prism layer stretching		1.100
Timestep	s	0.100

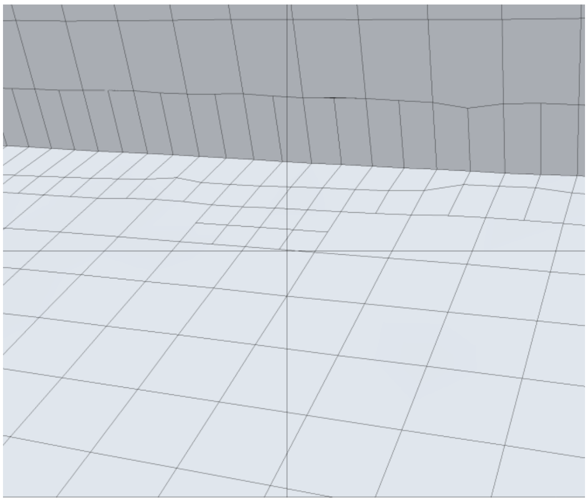

Figure 3. Trimmed hexahedral mesh changing mesh size to account for changes in geometry

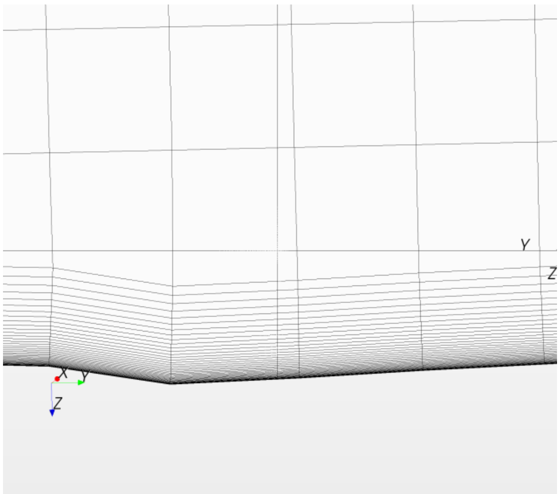

Figure 4. Prism layer creation: Refinement of mesh close to the floor of the model

2.3 BOUNDARY CONDITIONS

2.3.1 BOUNDARY CONDITION TYPE

Each surface of the model exterior must be classified as a boundary type that allows the model to perform calculations and take into account any known parameters at the boundaries. The following boundary types were assigned:

Table 2. Boundary Conditions

Model surface	Boundary type
Taranaki St Inlet	Velocity inlet
Arthur St Inlet	Velocity inlet
Buckle St Outlet	Pressure Outlet
Taranaki St Outlet	Pressure Outlet
Top of model	Pressure Outlet
Ground	Wall
Buildings	Wall

A velocity inlet boundary is where the flow velocity entering the model is known. The inflows to the model were set using a constant velocity distributed over an area stretching across the cross-sectional width of Arthur and Taranaki St respectively, with the water surface at a constant elevation. Approximate values for the flow depth and height were established using Manning's calculation, then scaled to obtain an inflow equal to that determined by the stormwater design calculation. The remainder of the surface at the inlet is assigned as air. A pressure outlet represents a boundary with a specified pressure, which was set to 0.0 Pa for this model.

2.3.2 FLOW DEVELOPMENT

The choice of appropriate boundary type is important so that a realistic flow regime may develop before the flow reaches the intersection. The Manning's equation was used to determine the likely flow depth and velocity of the inflow to the model. Four velocity profiles were taken within the model to show the development of the velocity profile as the water enters the model from Taranaki St. The locations of the velocity profiles are shown in Figure 5. The development of the velocity profile is shown in Figure 6. The figure shows that the velocity profile is not developed in the first profile at 1 m, as can be seen by the irregular shape of the curve, and slower flow velocity in the top of the water column. The flow is nearly developed at 5 m, and is fully developed at the 10 m profile.

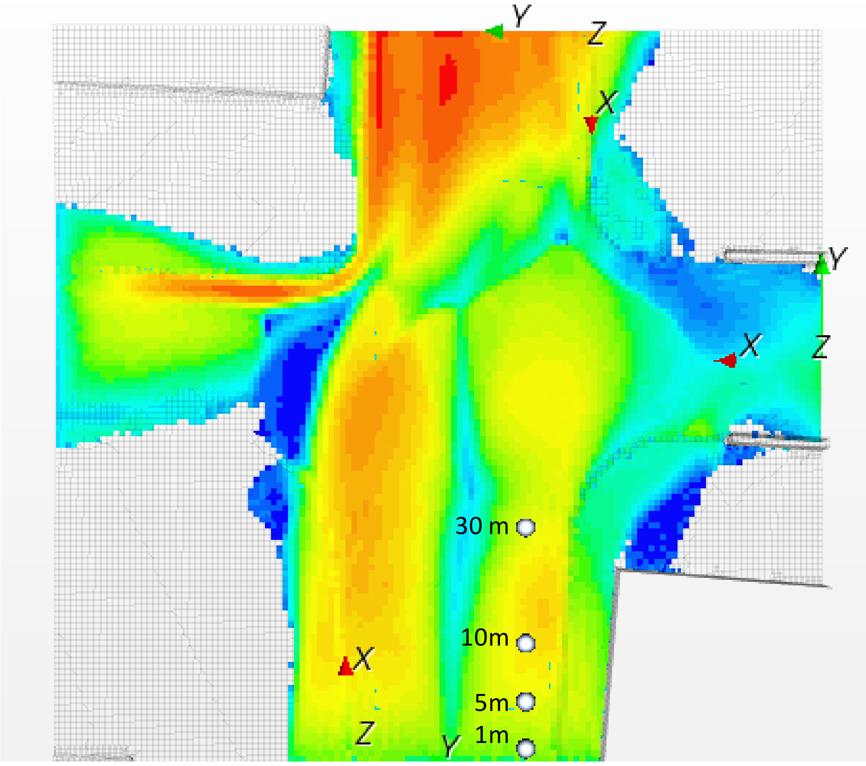

Figure 5. Location of velocity profile readings

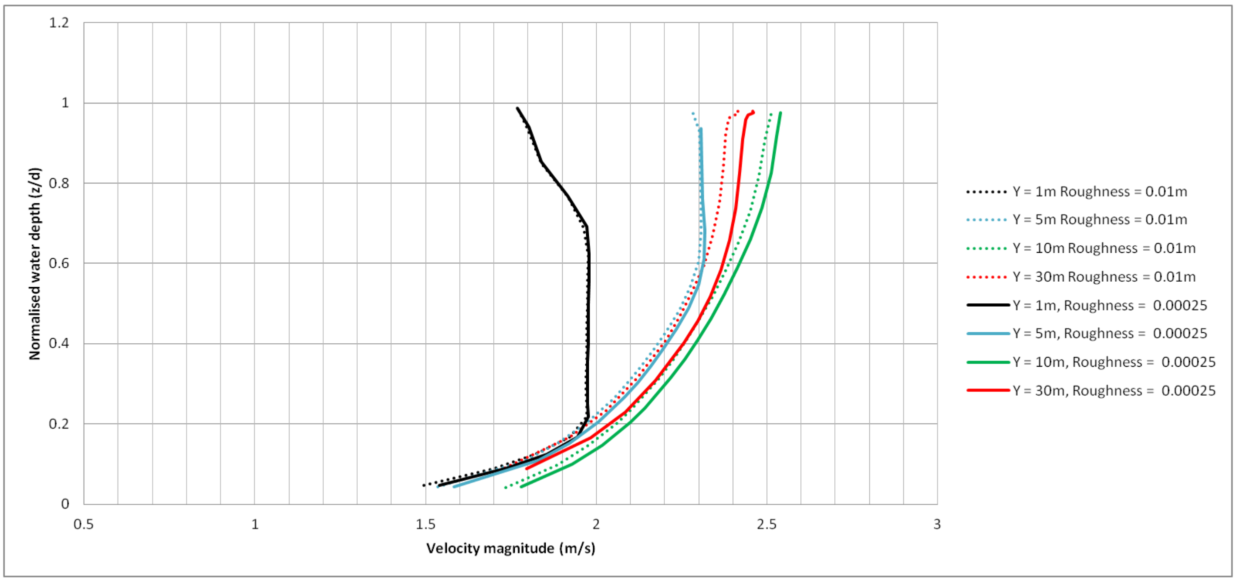

Figure 6. Velocity profiles taken at 1 m, 5 m, 10 m, and 30 m downstream from the Taranaki St model boundary. Profiles taken from the 500 yr ARI model, with floor roughness height at 0.01 m and 0.00025 m

2.3.3 ROUGHNESS

A roughness height was used to represent the road surface. The roughness affects the flow behavior in the transition between the laminar flow close to the wall and turbulent regions in the main body of the flow. The Moody diagram suggests roughness heights (ε) for the following surfaces as per Table 3. As an approximate upper limit, the maximum aggregate size for the road surface is approximately 10 mm. A sensitivity test between roughness height of 0.25 mm and 10 mm is shown in Figure 6. The effect of the change in roughness on the velocity profile is minor, with slightly increased velocities with the lower roughness height. The difference in outflow from the model down Buckle St between the two model runs was insignificant. Models were run with a roughness of 0.25 mm.

Table 3. Possible roughness values (Granger 1985)

Material	ε (mm)
Concrete, coarse	0.25
Concrete, new	0.025
Cast iron	0.15

2.4 MODEL SCENARIOS

The purpose of the model was to test the designed geometry of the underpass to evaluate the volume of water that would enter the underpass in a flood event. Two different underpass geometries were tested, where Geometry 2 features a reduction in elevation at the underpass entrance from the intersection by 60 mm to reduce the protrusion of the underpass roof into the park overhead.

The primary network capacity was calculated to conservatively accommodate 1 m^3/s. A detailed 1D model of the network was not available for use in the development of this model.

The design parameters for each of the modelled rainfall events are tabulated in Table 4. Both Geometry 1 and Geometry 2 have the same input parameters for their respective rainfall events.

Time constraints were a major issue in this project, requiring a fast model build and model runs. The required simulations were successfully completed within approximately 3 weeks.

Table 4. Input Parameters

	Parameter	Units	500 yr ARI	100 yr ARI	10 yr ARI
Taranaki St	**Inflow**	**m^3/s**	5.40	3.50	1.17
	Inflow velocity	**m/s**	1.66	1.08	0.36
	Maximum inflow depth	**m**	0.26	0.26	0.26
Arthur St	**Inflow**	**m^3/s**	0.33	0.23	0.13
	Inflow velocity	**m/s**	0.62	0.44	0.25
	Maximum inflow depth	**m**	0.12	0.12	0.12

2.5 ASSUMPTIONS AND RISK

The following assumptions were made in the construction of the model:

- The 3D mesh is generated from a combination of 0.05 m contours of the existing ground levels, and a triangulated network surface of the design intersection alignment. The real intersection overland flow will be subject to any changes made to the intersection design, and construction error.
- Roughness height of the intersection surface: 0.25 mm
- Inflows were based on the Rational Method
- Inflow volume is an approximation of what would be expected in a real flood
- Inflow velocity is constant
- No cars are present in the intersection at the time of flood

Rainfall entering the model boundaries was not included in the model, which is reasonable as the rainfall falling into the 0.005 ha model area would not significantly change the results.

As with all modelling, this numerical simulation is an approximation of natural phenomena, and is subject to simplification and approximations that introduce a level of uncertainty in the final result. To account for the inherent uncertainty in the numerical model simulation, a 30% error factor was applied. In addition to this, a 1.5 Factor of Safety was applied when using the results from this model in all subsequent design calculations to determine a design flow for the design of the underpass drainage.

3 MODEL OUTPUT

CFD is capable of extremely detailed analysis of flow properties. The design output requirement from this model was primarily the quantification of the peak flow rates of water entering the underpass. Therefore, the outputs generated from the CFD model are limited to flow rate, water velocity and water depth. Some results from the Geometry 1 simulations and the 100 and 10 year event simulations for Geometry 2 are excluded in this paper to avoid repetition.

Figure 7 shows the relationship between the flood event and the flow rate to the underpass. The three modelled storm events show an increase in flow to the underpass between the 10 year and 100 year events, and a further increase between the 100 and 500 year events. The model results show a clear increase in flow to the underpass with Geometry 2, this is expected as the intersection geometry has a reduced elevation at the underpass entrance from the intersection for Geometry 2.

Figure 8 shows the peak water depth of the flow at the model boundaries for the 500 yr ARI flood for Geometry 2. The peak water depth profiles show the distribution of water depth across the width of the road. The flow down Taranaki St is effectively diverted to the gutters, with a reduced depth of water travelling along the centre of the road. Flow into the underpass is concentrated toward the downhill (Taranaki St downstream) edge of the underpass. Figure 9 shows the peak water elevation of the flow exiting the model boundaries for the 500 yr ARI flood for Geometry 2. The water depth of the flow exiting Buckle St is small, and therefore the water elevation is dominated by the camber of the road. The water elevation of flow exiting Taranaki St is relatively level compared to the road geometry, the gutters of the road are overflowing, with water covering the road surface and spilling out of the gutter away from the road.

Figures 11 to 13 show the velocity vector schematics for all three modelled events for Geometry 2. Inspection of the velocity vector maps show that the flow behaviour is very responsive to the intersection geometry. As the flow approaches the intersection from Taranaki Street it is travelling in parallel streamlines, and is shallower toward the road centreline. As the flow enters the intersection the change to a constant 0.5% crossfall spreads the flow and diverts if towards Arthur Street and away from Buckle Street.

Flow from Arthur St enters the intersection and causes a flow eddy to occur at the southern corner between Arthur and Taranaki St. The impact of the Arthur St flow can be seen on the main flow path, as it causes water to backup as the flows from the two roads converge.

The momentum of the flow down Taranaki St carries much of the flow past the underpass entrance. This effect is particularly noticeable in the higher flood events. The main mechanism for flow entering the underpass is caused by flow hitting the kerb on the downhill corner between Taranaki St and the underpass. This results in a reduction in flow velocity in that area, and a divergence of some of the flow toward the underpass.

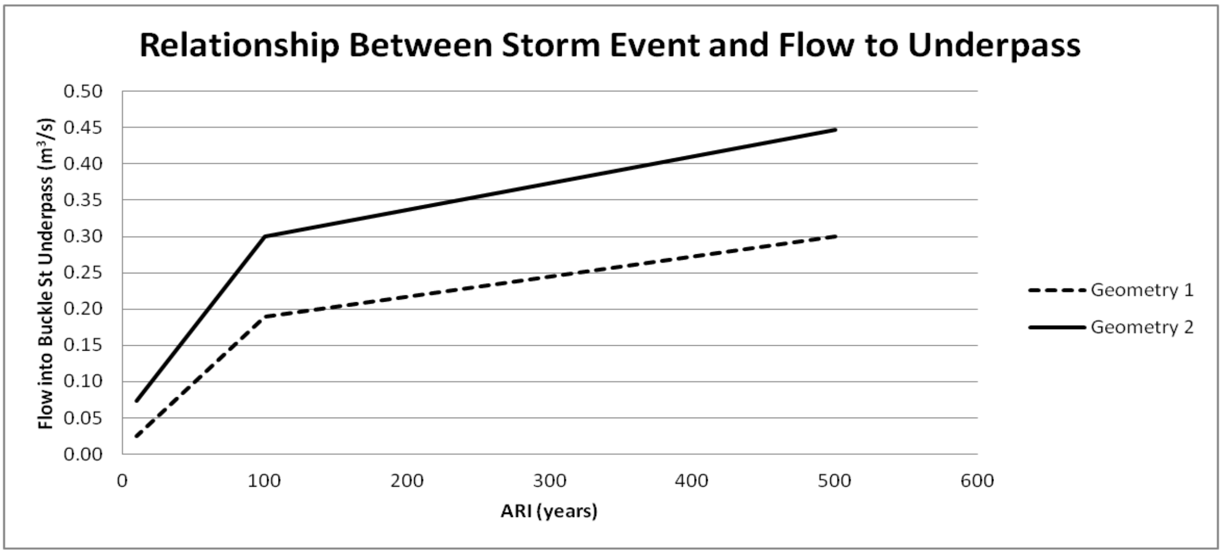

Figure 7. Relationship between storm event and peak flow to underpass

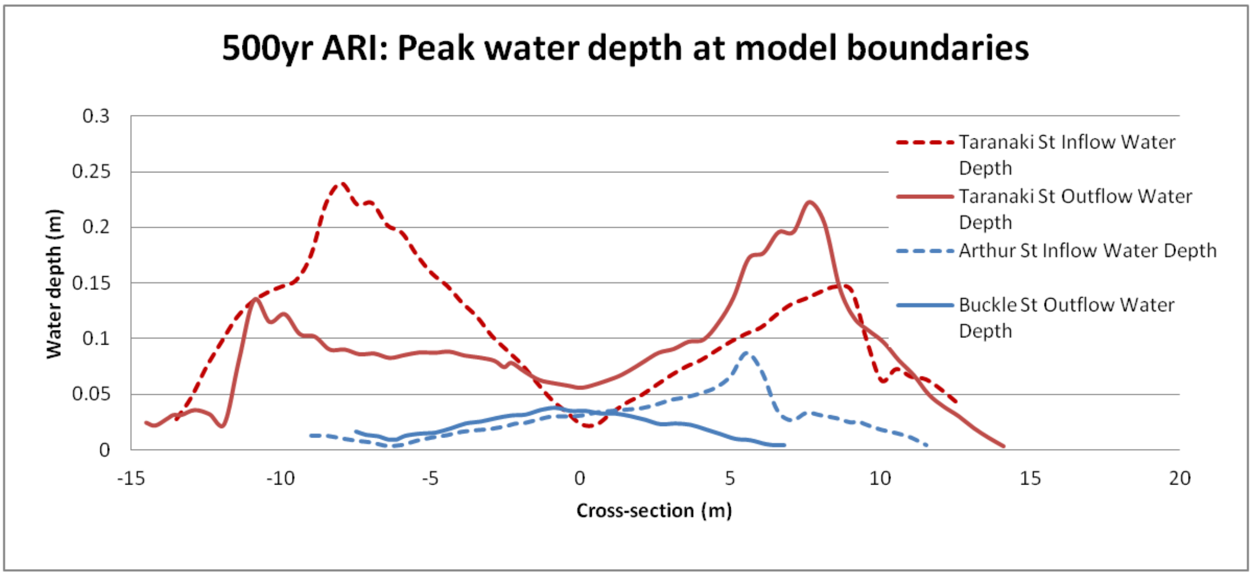

Figure 8. Geometry 2 500yr ARI peak water depth at model boundaries, see Figure 10 for cross section locations

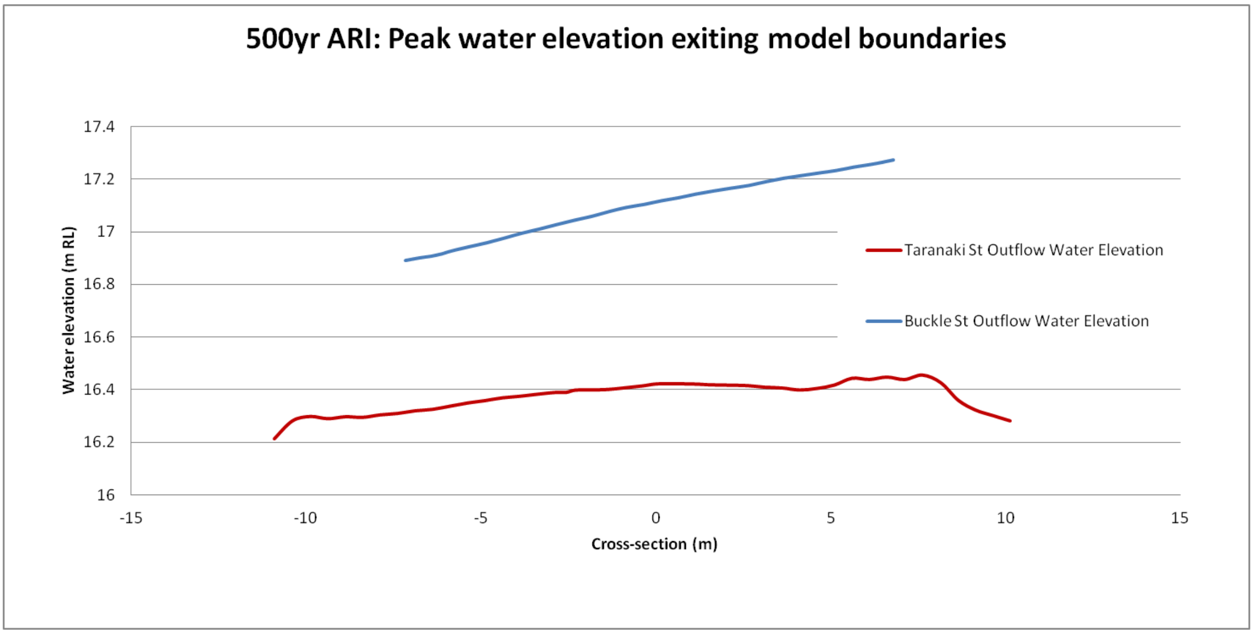

Figure 9. Geometry 2 500yr ARI peak water elevation of flow exiting model boundaries, see, Figure 10 for cross section locations

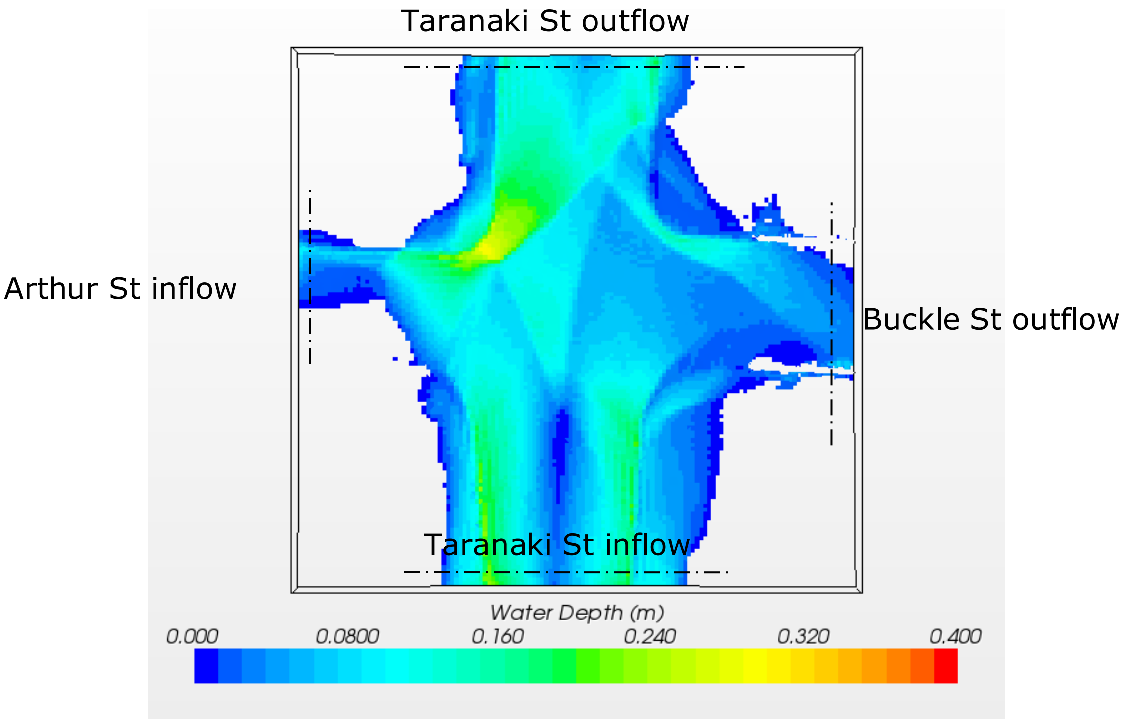

Figure 10. 500 yr ARI peak water depth, showing Figure 8 cross section locations

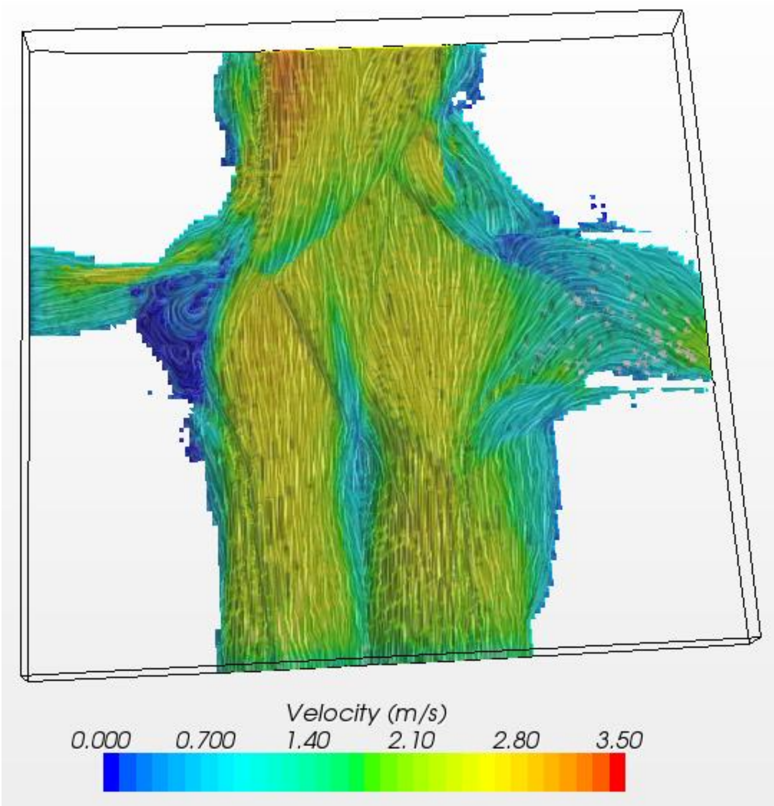

Figure 11. 500 yr ARI Velocity vector

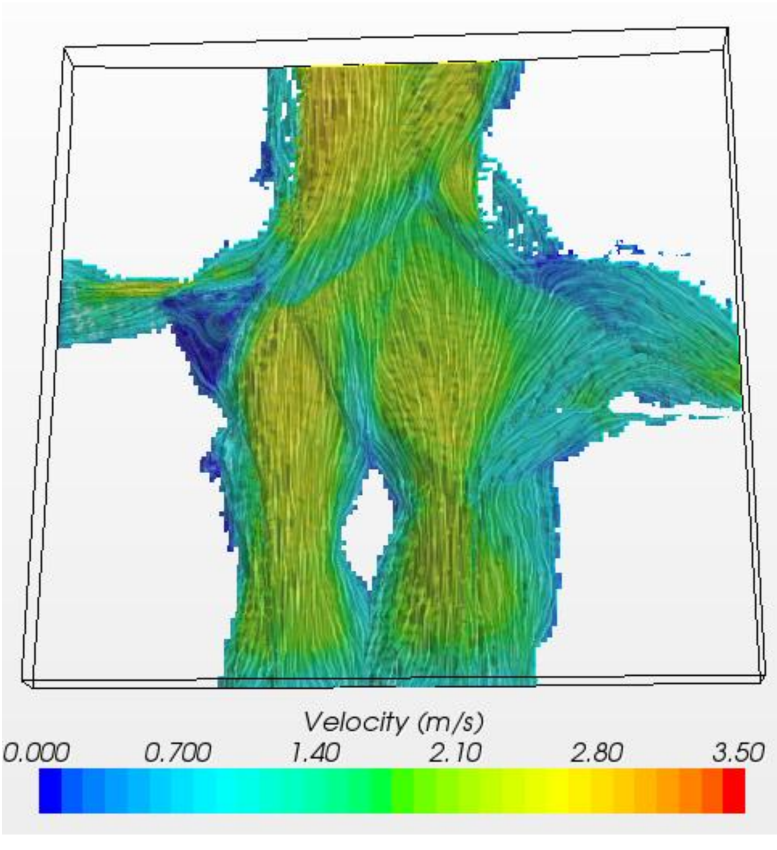

Figure 12. 100 yr ARI Velocity vector

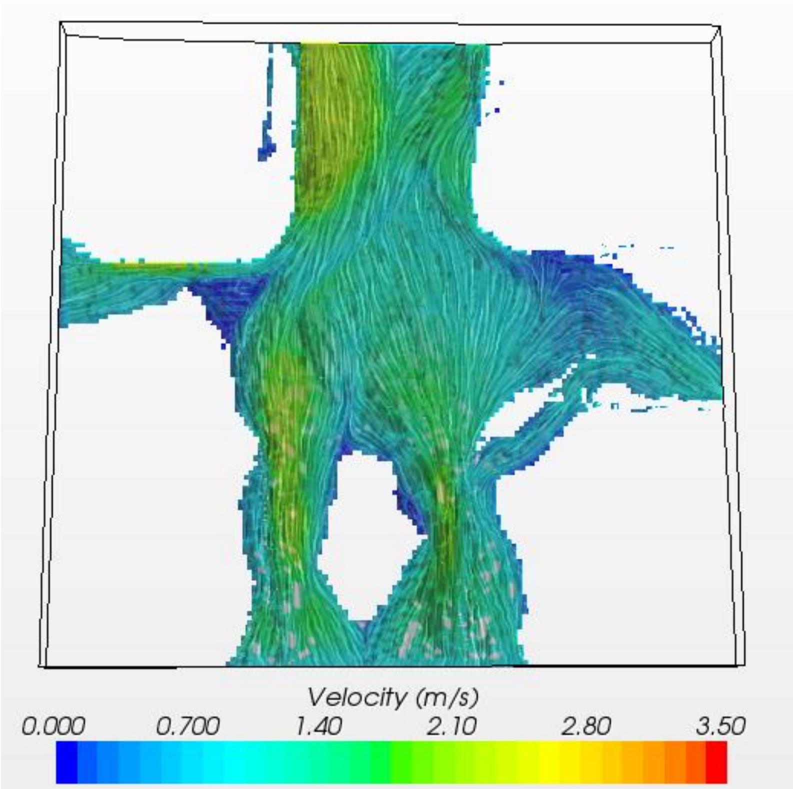

Figure 13. 10 yr ARI Velocity vector

3.1.1 ASSESSMENT OF OUTPUT

Mass balance within the model was calculated by summing the flow into and out of the model on all boundaries. Mass balance is a common method for checking that the model is working correctly. All models had a mass balance error less than the typically acceptable error of 5%. This indicates that negligible water in the model was unaccounted for. The error that was observed was primarily through flow exiting the model through the top boundary. This is most likely due to diffusion of the water into the air. This phenomenon occurs in places where instabilities arise in the model, such as where sharp changes in ground surface cause unusually high velocities. Care was taken to minimise diffusion, but it is difficult to eliminate diffusion completely.

Evaluation of "Wall Y+" values is one method of verifying that the model is correctly configured. Wall Y+ values between 35 and 150 indicate that flow properties in the boundary layer are conforming to the law of the wall. Not all of the flow will have values within this range, however if the main flow column is within the acceptable range it is a good indication that the model is behaving within the design constraints. Figure 14 shows the Y+ values in the bulk of the flow are within the acceptable range.

Without calibration of the model with a physical study it is difficult to quantify the limits of error of the model results. Although widely applied to mechanical engineering problems, the use of CFD in modelling overland flow is less widely published. Considering this uncertainty in the accuracy with which the CFD model represents the actual case, it is estimated that a potential level of error in the underpass flow estimation is 30%.

It would be of value to compare the results from this model with another method. This would build confidence in the application of CFD to this scenario, and would provide an objective gauge in the level of error that should be expected. A 2D hydraulic model is planned for the near future as a verification study.

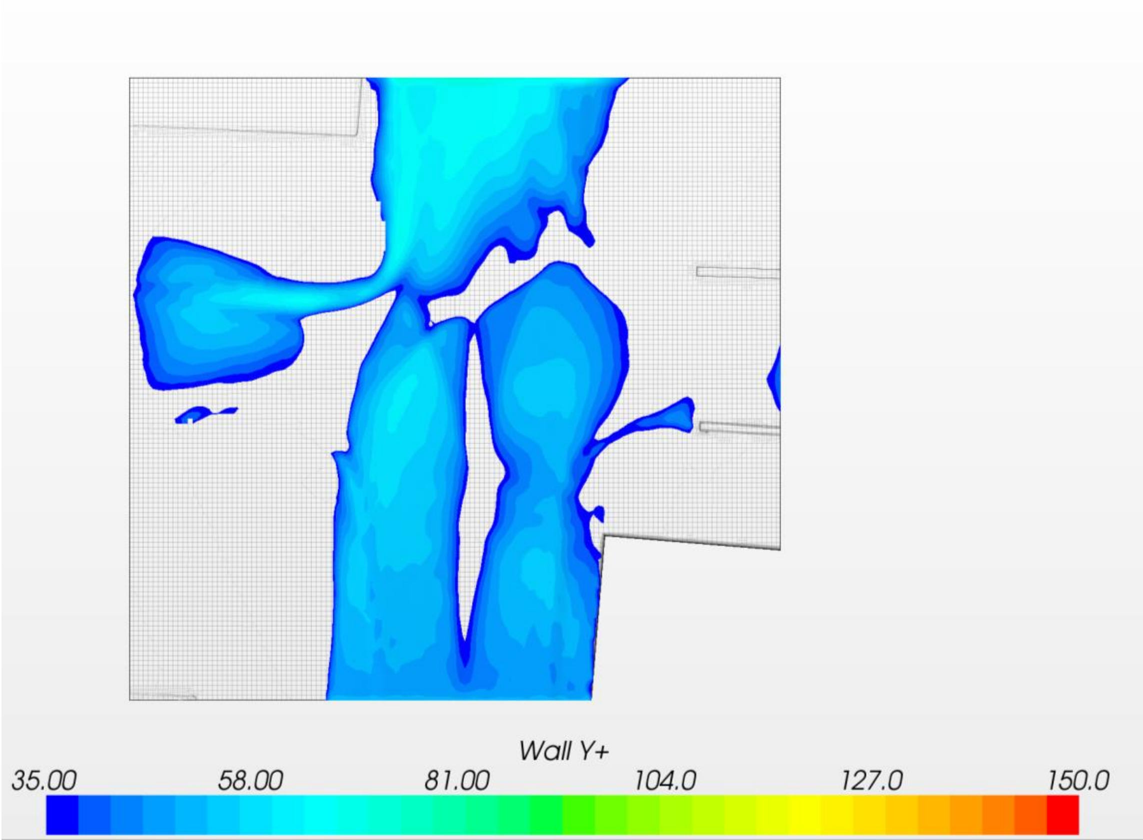

Figure 14. Wall Y+ plot

4 SUMMARY AND CONCLUSIONS

Uncertainty around the flow of water entering the Buckle St underpass was identified, due to the complicated geometry of the intersection. CFD modelling was undertaken to refine the intersection surface design, to assist in striking the best balance between a safe road alignment and flood protection. A CFD model was built to simulate overland flow down Taranaki St and the potential for this flow to enter the Buckle Street underpass. Three flood events were modelled, which were the 10 yr ARI, 100 yr ARI and 500 yr ARI. Two intersection geometries were tested to evaluate the sensitivity of the geometry to increases in flow to the underpass.

In each simulation, the flow rate of water exiting the model to the Buckle St underpass was quantified. The mass balance error for all models was under 5%. When applying the model results in design, an error of 30% was applied to each outflow value to account for uncertainty in the accuracy of the numerical model. Additionally, a 1.5 factor of safety was applied when using the model results in design calculations.

The CFD model was a useful design tool in simulating the three dimensional flows that could be expected in a major flood event. The model indicated that flow to the underpass can be expected in all floods greater than the 10 yr ARI. The model results influenced the design of the intersection geometry, and informed the project designers allowing them to strike the best balance between a safe road alignment and flood protection. The model results were used by the MPA stormwater team in the design of the underpass drainage to maintain underpass flooding at acceptable levels in extreme events.

A future study is planned to use a 2D hydraulic modelling package to evaluate the same scenario as a verification of the CFD results.

ACKNOWLEDGEMENTS

The Authors of this study would like to thank the help provided by Paul Bosauder and the team from Matrix Computing. The authors would also like to thank our client NZTA for their support. Finally, the authors would like to acknowledge the rest of the team members in the Memorial Park Alliance.

REFERENCES

CD-Adapco (2013) 'Star-CCM+ Training Manual' https://support.cd-adapco.com

Gräf, L. and Kleiser, L. (2011) 'Large-Eddy simulation of double-row compound-angle film cooling: Setup and validation.' *Computers and Fluids*, 43, 1, 58-67

Granger, R. A. (1985) 'Fluid Mechanics' 498

Jain, M., Puranik, B., and Agrawal, A. (2011) 'A numerical investigation of effects of cavity and orifics parameters on the characteristics of a synthetic jet flow.' *Sensors and Actuators A: Physical,* 165, 2, 351-366

Khan, S., Melville, B., Shamseldin, A. Y., Fischer, C. (2013) 'Investigation of Flow Patterns in Storm Water Retention Ponds using CFD.' *Journal of Environmental Engineering*, 139, 1, 61-69

Shilton, A., Kreegher, S., and Grigg, N. (2008) 'Comparison of computation fluid dynamics simulation against tracer data from a scale model and full-sized waste stabilization pond' *Journal of Environmental Engineering,* 134, 10, 845-850

Toombes, L. Chanson, H. (2011) 'Numerical Limitations of Hydraulic Models' 34th IAHR World Congress, 26 June – 1 July 2011, Brisbane, Australia

Vakili, M. H. and Esfahany, M. N. (2009) 'Cfd analysis of turbulenc in a baffled stirred tank, a three-compartment model.' *Chemical Engineering Science*, 64, 2, 351-362

Vega, G. P., Pena, M. R., Ramirez, C., and Mara, D. D. (2003) 'Application of CFD modelling to study the hydrodynamics of various anaerobic pond configurations' *Water Science & Technology*, 48, 2, 163-171

WCC (2012) WCC Regional Standard for Water Services Sept 2012 (Draft)

Wu, B. (2011) 'CFD investigation of turbulence models for mechanical agitation of non-Newtonian fluids in anaerobic digesters.' *Water Research*, 45, 5, 2082-2094