

APPENDIX A

IMAGES OF THE NOVEMBER 1984 FLOOD



Photograph No. 1: Looking upstream across The Avenue.



Photograph No. 2: Looking north along Lakeside Road.



Photograph No. 3: Looking west along Hillview Road.



Photograph No. 4: Looking north at the western entrance to Eastwood railway station.



Photograph No. 5: Commuters (including schoolgirls) wading through floodwaters in the railway pedestrian system.



Photograph No. 6: Looking south along Railway Parade.

APPENDIX B

DESCRIPTION OF TUFLOW MODEL SOFTWARE

B1 BACKGROUND

B1.1 Introduction to Floodplain Modelling

Floodplains are hosts to industrial sites, urban and rural communities, and environmentally sensitive areas. During periods of flooding, most damage and disruption will occur on the floodplains – not in the creeks and rivers. Therefore, correctly modelling floodplains and their interaction with rivers is very important.

Different modelling methods can be applied according to the floodplain's hydraulic characteristics and the study objectives and resources. The simpler methods lump the left and right bank floodplains with the river in a one-dimensional (1D) representation. This computation approach is fast, however, there are limitations such as the floodplain flood level being always assumed to be the same level as the river level.

Alternatively, there is the more detailed approach of modelling the river and adjacent floodplains as separate flow paths. This method increases model complexity, development and running time, and requires greater human and computer resource. The currently most advanced, yet still economical, method to undertake floodplain modelling is using two-dimensional (2D) discretisation.

A detailed 2D approach is recommended in areas where significant differences between river and floodplain flood levels and separate flow paths occur, especially if these differences result from a management option. Therefore a 2D detailed approach is particularly relevant for the Terrys Creek Catchment, which incorporates many overland flowpaths and floodplain flows.

B1.2 Terrys Creek Model Configuration

The modelling software TUFLOW was used to set up a hydrodynamic, dynamically linked 2D/1D hydraulic model of the catchment. The model is a mixture of 1D and 2D domains with the 2D domain covering the whole catchment for predicting floodplain and overland flowpath flow behaviour in floods. The 1D domain is suited for pipe flow modelling where the flow is unidirectional. Dynamic links exist between the 2D domain and the 1D domain at the location of the stormwater drainage pits and between the 2D domain and the 1D open channel sections of the lower reaches of Duck River.

B2 TUFLOW

B2.1 Overview

TUFLOW solves the full 2D shallow water equations based on the scheme developed by Stelling (1984). The solution is based around the well-known ADI (alternating

direction implicit) finite difference method. A square grid is used to define the discretisation of the computational domain. TUFLOW also has the ability to be dynamically linked to 1D models and have 1D models dynamically nested inside or through the 2D domain.

Improvements to the Stelling 1984 scheme, including a robust wetting and drying algorithm and greater stability at oblique boundaries, and the ability to dynamically link a quasi-2D model were developed by Syme (1991). Further improvements including the insertion of 1D elements (channel, pipe, weir) inside a 2D model and the modelling of constrictions on flow such as bridges and large culverts, and automatic switching into and out of upstream controlled weir flow have been developed subsequently (WBM, 2000).

Hydraulic structure flows through large culverts and bridges are modelled in 2D and include the effects of bridge decks and submerged culvert flow. Flow over roads, levees, bunds, etc is modelled using the broad-crested weir formula when the flow is upstream controlled. For smaller hydraulic structures such as pipes or for weir flow over a bridge, 1D models can be inserted at any points inside the 2D model area.

B2.2 Floodplain Modelling Equations

The shallow water equations are the equations of fluid motion used for modelling long waves such as floods, ocean tides and storm surges. They are derived using the hypotheses of vertically uniform horizontal velocity and negligible vertical acceleration (i.e. a hydrostatic pressure distribution). These assumptions are valid where the wave length is much greater than the depth of water.

The 2D shallow water equations in the horizontal plane are described by the following partial differential equations of mass continuity (Equation B.1) and momentum conservation in the X and Y directions (Equations B.2a and b) for an in-plan cartesian coordinate frame of reference.

$$\frac{\partial \zeta}{\partial t} + \frac{\partial(Hu)}{\partial x} + \frac{\partial(Hv)}{\partial y} = 0 \quad (\text{B.1})$$

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} - c_f v + g \frac{\partial \zeta}{\partial x} + g u \frac{\sqrt{u^2 + v^2}}{C^2 H} - \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) = F_x \quad (\text{B.2a})$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + c_f u + g \frac{\partial \zeta}{\partial y} + g v \frac{\sqrt{u^2 + v^2}}{C^2 H} - \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) = F_y \quad (\text{B.2b})$$

where

ζ = Water surface elevation

u and v = Depth averaged velocity components in X and Y directions

H = Depth of water

t = Time

x and y = Distance in X and Y directions

c_f = Coriolis force coefficient

C = $Ch\zeta_{zy}$ coefficient

μ = Horizontal diffusion of momentum coefficient

F_x and F_y = Sum of components of external forces in X and Y directions

The terms of the equations can be attributed to different physical phenomena. These are propagation of the wave due to gravitational forces, the transport of momentum by advection, the horizontal diffusion of momentum, and external forces such as bed friction, rotation of the earth, wind, wave radiation stresses, and barometric pressure.

The 2D shallow water equation scheme adopted incorporates all of the above physical processes. External forces such as wind, wave radiation stresses, and barometric pressure are incorporated into the code but are not used in this study.

For further information on the 2D solutions, refer to Syme 1991.

B2.3 Open Channel Flow 1D Modelling Equations

TUFLOW uses an explicit finite difference, second-order, Runge-Kutta solution technique (Morrison and Smith, 1978) for the 1D equations of continuity and momentum as given by Equations B3 and B.4. The equations contain the essential terms for modelling periodic long waves in open channels, that is: wave propagation; advection of momentum (inertia terms) and bed friction (Manning's equation).

$$\frac{\partial(uA)}{\partial x} + B \frac{\partial \zeta}{\partial t} = 0 \quad (\text{B.3})$$

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + g \frac{\partial \zeta}{\partial x} + k |u| u = 0 \quad (\text{B.4})$$

where

u = depth and width averaged velocity

ζ = water level

t = time

x = distance

A = cross sectional area

B = width of flow

k = energy loss coefficient = $\frac{gn^2}{R^{4/3}}$

n = M'annings n

R = Hydraulic Radius

g = acceleration due to gravity

The spatial discretisation of an area of interest is carried out as a network of interconnected nodes and channels. The nodes represent the storage characteristics of the open channel, while the channels model the hydraulic conveyance characteristics.

The continuity equation is solved at the nodes, while the momentum equation is solved for the channels. The output consists of water levels at the nodes, and flows, velocities and integral flows (flow integrated over time) at the channels.

For further information on the 1D solutions, refer to Syme 1991.

B2.4 Pipe Flow Modelling Equations

In pipe flows, the area of flow is fixed, and for a known flow in a given size of conduit the velocity can be calculated directly. The energy equation for the total flow in a pipe can be expressed as:

$$\frac{V_1^2}{2g} + h_1 + z_1 = \frac{V_2^2}{2g} + h_2 + z_2 + \Delta H$$

where ΔH is the energy loss in the pipeline between the two sections 1 and 2. The energy lost through turbulence is caused by two mechanisms:

- The drag of the pipe walls on the flow. This mechanism is known as the 'friction' loss; and
- Turbulence generated wherever there is a change to the direction and/or magnitude of flow. This mechanism is known as the 'form' loss.

The friction losses are continuous over the length of a pipeline; the form losses are localised in the immediate vicinity of the element causing the energy loss.

The loss of energy due to hydraulic resistance of a pipe is a function of the velocity of the flow, V , the internal pipe diameter, D , the length of the pipe, L , and the roughness of the pipe internal surface. There are several empirical formulae for the calculation of friction losses in pipes that have been derived through research. The Manning's equation is appropriate for use when the flow is in the fully-turbulent range, which is the case in rough conduits and at high flows in stormwater mains.

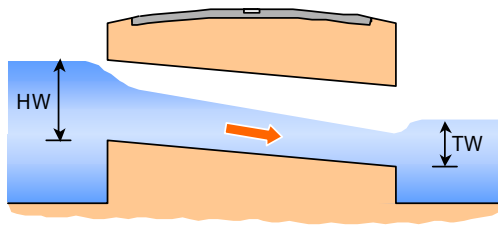
TUFLOW uses the Manning's equation to model pipe flows. For circular pipe, the equation can be written as follow:

$$V = \frac{0.397 D^{2/3} \sqrt{S_f}}{n}$$

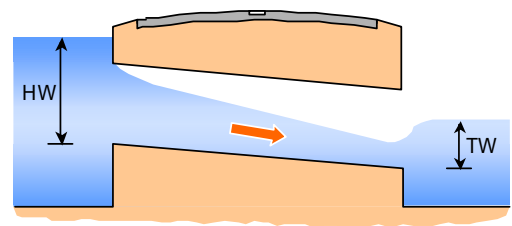
where S_f is the hydraulic gradient and n is the roughness parameter. Although n is a function of the conduit size, it is relatively insensitive to the pipe diameter and is assumed constant in the TUFLOW calculations.

When the pipes are not flowing full, ten flow regimes are possible within the TUFLOW software as illustrated in Figure B1 and Figure B2. Upstream water levels are calculated from the 1D equations and/or standard culvert discharge relationships.

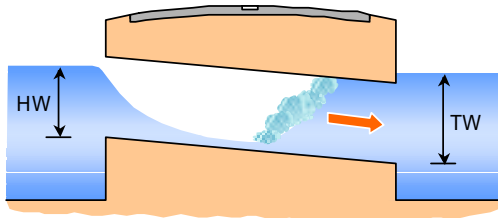
INLET CONTROL FLOW REGIMES



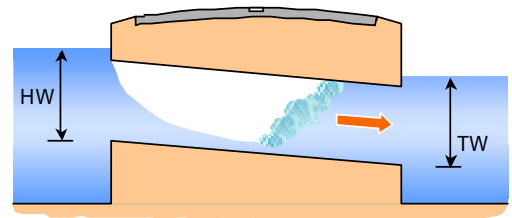
**A: Unsubmerged Entrance,
Supercritical Slope**



**B: Submerged Entrance,
Supercritical Slope**



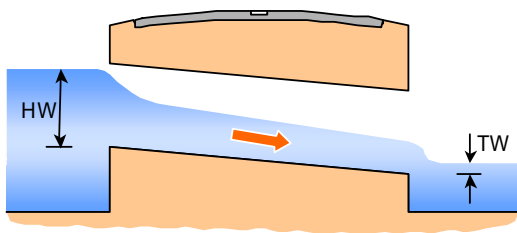
**K: Unsubmerged Entrance,
Submerged Exit
Critical at Entrance**



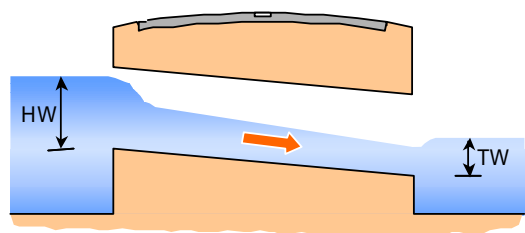
**L: Submerged Entrance,
Submerged Exit
Orifice Flow at Entrance**

Figure B1 *1D Inlet Control Culvert Flow Regimes in TUFLOW*

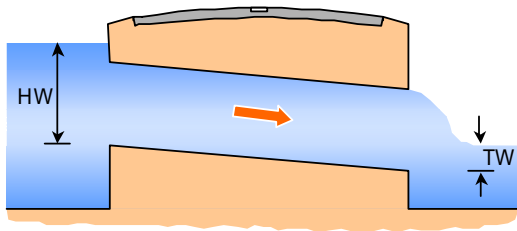
OUTLET CONTROL FLOW REGIMES



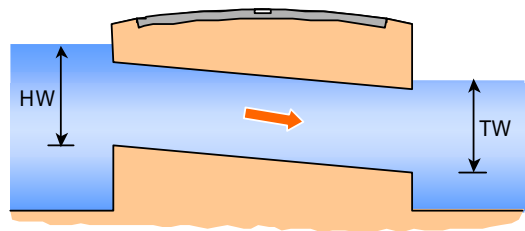
C: Unsubmerged Entrance, Critical Exit



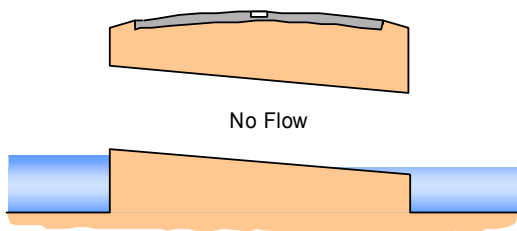
D: Unsubmerged Entrance, Subcritical Exit



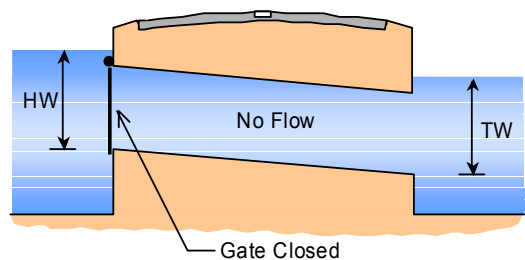
E: Submerged Entrance, Unsubmerged Exit



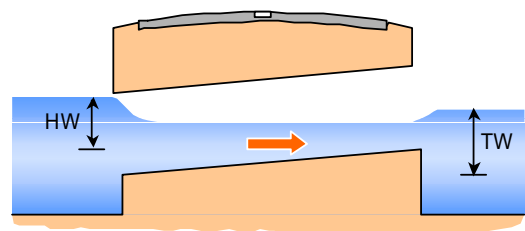
F: Submerged Entrance, Submerged Exit



**G: No Flow
Dry or Flap-Gate Closed**



H: Adverse Slope, Submerged Entrance



**J: Adverse Slope, Unsubmerged Entrance
(Critical or Subcritical at Exit)**

Figure B2 1D Outlet Control Culvert Flow Regimes in TUFLOW

B2.5 Stormwater Drainage Network Modelling

The coupling between the floodplain modelling and the pipe flow modelling occurs at the location of the stormwater drainage network pits (see Figure B3). The computation calculates a flow rate between the 2D floodplain cell and the 1D pipe based on:

- Overland water level in the floodplain;
- Energy level in the pipe at the location of the pit; and
- Lintel opening dimensions.

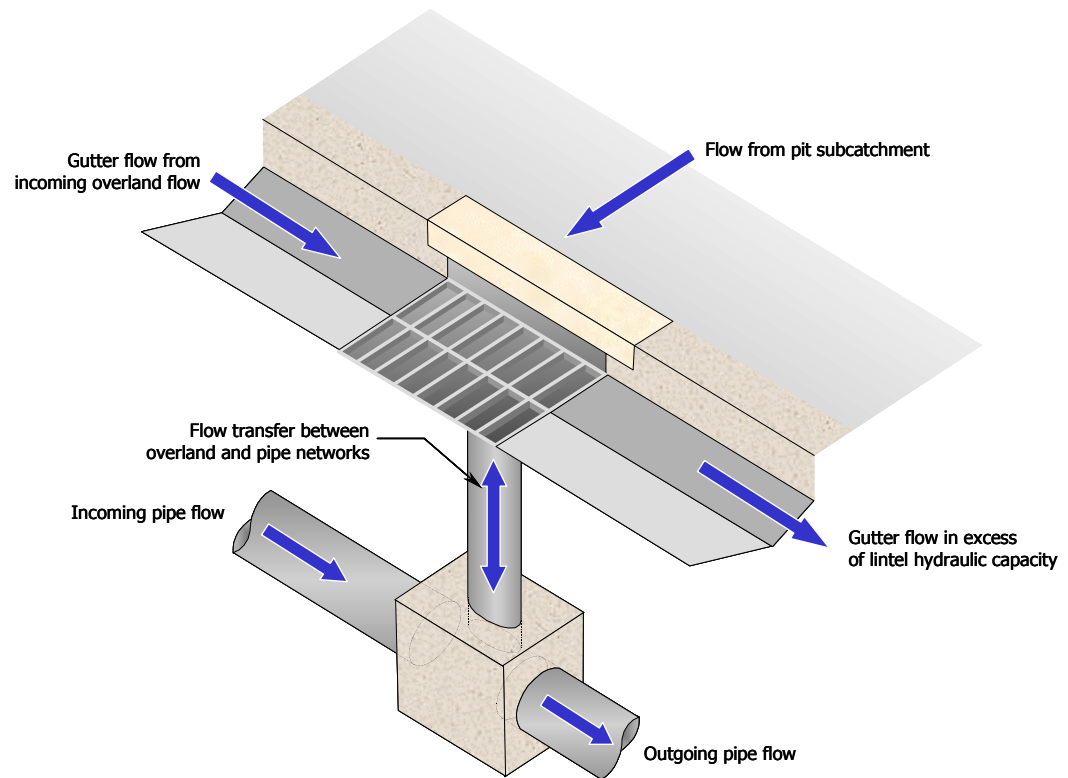
The modelling link representing the pit acts essentially as a zero-length culvert. The actual lintel clear opening dimensions have been used to represent the lintel as a zero-length box culvert with widths equal to the widths of the lintel and a standard height of 0.095m.

Representing the lintels as box culverts allows 11 flow regimes (as coded into TUFLOW) to simulate the culvert flow. The most common flow regimes would be:

- Un-submerged entrance, super critical flow, inlet control;
- Submerged entrance, super critical flow, inlet control; and
- Submerged entrance, submerged outlet, outlet control.

The representation of storage in the pit structure is approximated by the volume of a 1m long box culvert with the lintel opening area cross-section.

The 2D model domain has some limitations in representing the exact depth of flow at a lintel in a kerb. This is due to the resolution of the model (3m grid). For the smaller frequent events (e.g. 1 year and 2 year ARI events), this error is likely to be proportionally larger. Hence, the ability of the TUFLOW model to accurately represent the interaction between overland flow and inlet flow is somewhat limited for the smaller events. Therefore, the results for these events are likely to over-estimate the flow along the overland flow system and under-estimate the flow in the pipe system.



Hydraulic Process of Pit Location

Figure B3: Drainage Pit

B3. TUFLOW Modelling Issues for Terrys Creek

B3.1 2D Domain

The 2D domain of the TUFLOW model is based on a 3m square grid. Each square grid element contains information on ground topography sampled from the DEM at a 1.5m spacing and surface resistance to flow (Manning's n value). The 3m grid cell size is adequate for the study area floodplain, as it is sufficiently fine to represent the variations in the floodplain topography and vegetation cover.

B3.2 1D Channels

The open channels and creeks are modelled using 1D elements. The 1D domain consists of cross-section nodes and channels that calculate the hydraulic conveyance and the storage within the open channels. Dynamic links exist between the 2D domain and the 1D domain along the boundary of the 1D channels. A schematic representation of the linking mechanism between the 1D domain and the 2D domain in the study area is presented in Figure B4.

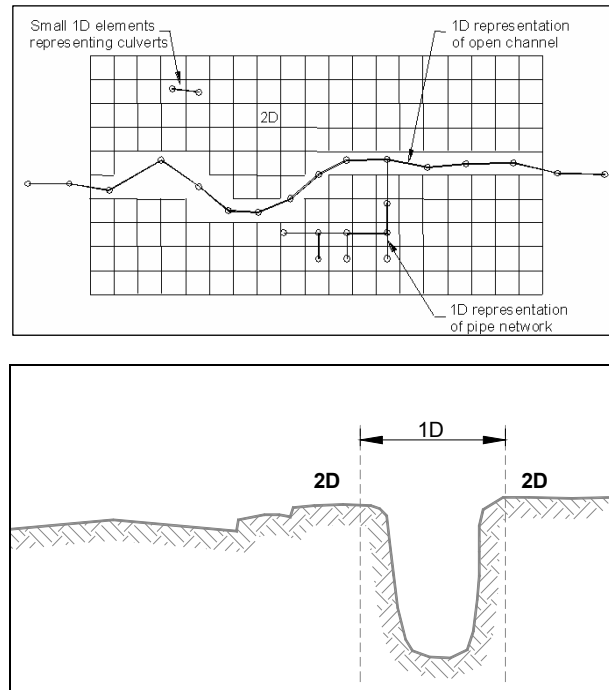


Figure B4 1D/2D Linking Mechanism

The 1D model is based on field data and cross-sections that were extracted from previous models and survey plans of the stormwater open drains (design and/or as constructed). The location and the number of cross-sections provide a sufficient description of the geometric variations of the river channel along its course. Model cross-sections are placed at locations of channel cross-section shape changes, upstream and downstream of flow constrictions and hydraulic structures, and at locations of longitudinal bed slope changes. It is not necessary to input several cross-sections along a uniform straight channel. A 1D channel is associated in the model with each cross-section. The cross-section shape and roughness is processed to determine conveyances with depths. A 1D node is created at each end of the 1D channel to represent the channel storage.

Additional 1D elements represent the bridges crossing the creek. Hydraulic structures in the 1D domain are modelled by replacing the momentum equation with standard equations describing the flow through the structure. The basic structures available are listed below:

- Bridges;
- Culverts; and
- Weirs.

The bridge opening cross section is described in the same manner to a normal channel. The highest level given in the cross-section data table is assumed to be the underside of the bridge deck, enabling the program to compute a correction for submerged decking. Bridge structures are modelled using a height varying form loss

coefficient. The coefficients are obtained from publications such as “Hydraulics of Bridge Waterways” (US FHA 1973). They include loss coefficients for bridge opening ratio, for piers, for eccentricity and for skewness and an automated option for applying losses once the deck becomes surcharged.

B3.3 1D Pipes

The pits and pipes network data was provided by the Council and compiled by the consultant. The data is intended to correspond to the entire pipe network. It was compiled from two different sources:

- Council asset database; and
- Site inspections.

Where invert levels of pits and pipes were missing, these levels were estimated based on typical cover types observed in adjacent areas.

B3.4 Roughness of Floodplain and Creek

The roughness of the creek and floodplain is represented in the model using the Manning’s roughness coefficient, n . The choice of the Manning’s n roughness values is made using engineering standards (e.g. Ven Te Chow, Arcement and Schneider) that have defined empirical values for specific ground cover types. The landuse analysis identified 17 different ground surface types with hydraulic significance in the catchment.

TABLE B1 TUFLOW Model Manning’s n Roughnesses

TUFLOW Material	Surface Type (Material)	Manning’s n
1	Urban – fences and typical gardens, backyards	0.1
2	Urban – units and strata titled land	0.025
4	Roads and paved/concrete areas	0.02
5	Short grass / bare earth	0.03
5	Vegetated area	0.05
6	Vegetated floodplain	0.08
8	Buildings	20
24	Channel (floodplain)	0.13

In the TUFLOW model 1D domain, the following roughnesses were adopted:

- Pipes: 0.016, corresponding to old concrete; and
- Channels: 0.016.

B3.5 Hydraulic Loss Coefficients at Structures

Pipeline fittings and changes in channel geometry generate head losses along the flowline. The calculation of losses is made as a function of the velocity head:

$$\Delta H = K \frac{V^2}{2g}$$

For pipes, the calculation uses the velocity associated with the flow rate. It is more complicated with open channel losses as it is often not possible to use a single, standard velocity as in the case of a pipe of constant diameter. TUFLOW uses the average velocity inside the structure (bridge, culvert) in the equation.

Values of K are almost entirely empirical but there have been extensive experimental measurements on standard fittings and bridges on which estimates can be based. The values adopted for the TUFLOW model follow recommended values. The TUFLOW headloss coefficients are presented in Table B2.

TABLE B2: TUFLOW Model Headloss Coefficients

Coefficient Description		Values
Structure ⁽¹⁾ Inlet Control Loss Coefficient	Circular	1.0
	Rectangular – Height	0.6
	Rectangular – Width	0.9
Structure ⁽¹⁾ Outlet Control Loss Coefficient	Entry	0.5 ²
	Exit	1 ²

(1): Bridges, culverts or pipes

(2): These are default values only. Actual values used in Terrys Creek were based on site specific assessments.

An additional feature allows the energy losses, associated with the contraction and expansion of flow lines into and out of a structure, to be automatically adjusted according to the approach and departure velocities in the upstream and downstream channels. The entrance and exit losses are adjusted according to the following equations:

$$K_{entrance_adjusted} = K_{entrance} \left[1 - \frac{V_{approach}}{V_{structure}} \right]$$

$$K_{exit_adjusted} = K_{exit} \left[1 - \frac{V_{departure}}{V_{structure}} \right]^2$$

TUFLOW can also introduce unadjusted bend or additional losses.