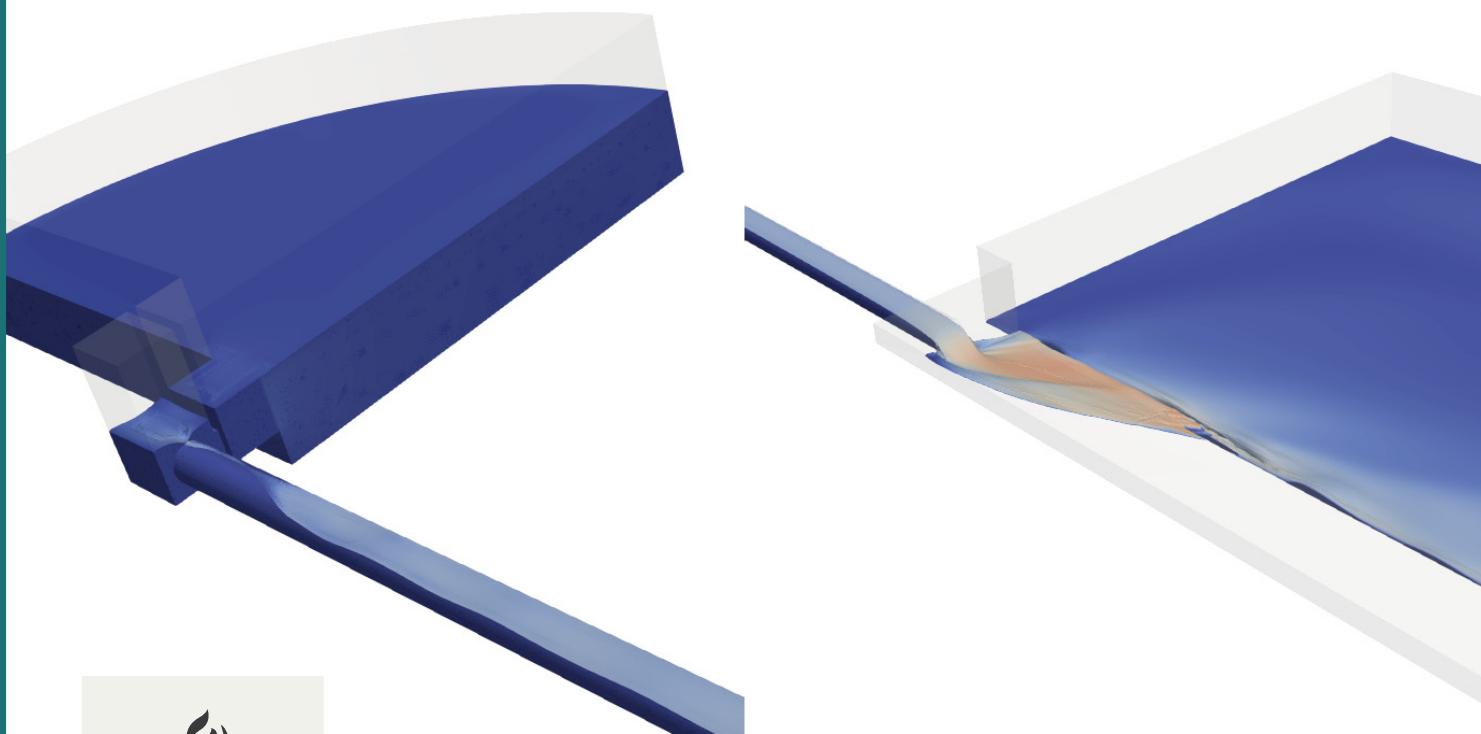




Flanders
State of
the Art

18_134_1
FHR reports

Side Drainage Operation in Dessel-Schoten Canal's Lock No. 3

Hydraulics and Navigability Assessment

DEPARTMENT
**MOBILITY &
PUBLIC
WORKS**

www.flandershydraulicsresearch.be

Side Drainage Operation in Dessel-Schoten Canal's Lock No. 3

Hydraulics and Navigability Assessment

Lopez Castano, S.; Verelst, K.; Verwilligen, J.; van Hoydonck, W.; Vercruysse, J.; Mostaert, F.



Legal notice

Flanders Hydraulics Research is of the opinion that the information and positions in this report are substantiated by the available data and knowledge at the time of writing.

The positions taken in this report are those of Flanders Hydraulics Research and do not reflect necessarily the opinion of the Government of Flanders or any of its institutions.

Flanders Hydraulics Research nor any person or company acting on behalf of Flanders Hydraulics Research is responsible for any loss or damage arising from the use of the information in this report.

Copyright and citation

© The Government of Flanders, Department of Mobility and Public Works, Flanders Hydraulics Research, 2021

D/2021/3241/019

This publication should be cited as follows:

Lopez Castano, S.; Verelst, K.; Verwilligen, J.; van Hoydonck, W.; Vercruysse, J.; Mostaert, F. (2021). Side Drainage Operation in Dessel-Schoten Canal's Lock No. 3: Hydraulics and Navigability Assessment. Version 3.0. FHR Reports, 18_134_1. Flanders Hydraulics Research: Antwerp

Reproduction of and reference to this publication is authorised provided the source is acknowledged correctly.

Document identification

Customer:	de Vlaamse Waterweg		Ref.:	WL2021R18_134_1
Keywords (3-5):	Drainage Structure, Computational Fluid Dynamics (CFD), flow assessment, lock entry/exit			
Knowledge domains:	Hydraulic structures – Culverts – Numerical Modelling Hydraulic Structures – Locks Hydraulics and Sediment – Hydrodynamics – Current Velocities and patterns – Numerical Modelling			
Text (p.):	33		Appendices (p.):	5
Confidential:	No	☒ Available online		
Author(s):	Lopez Castano, S.			

Control

	Name	Signature
Revisor(s):	Verelst, K.; Verwilligen, J.; van Hoydonck, W.	Getekend door: Kristof Verelst (Signature) Getekend op: 2021-02-11 17:56:42 +01:00 Reden: Ik keur dit document goed Getekend door: Jeroen Verwilligen (Signature) Getekend op: 2021-02-12 10:42:42 +01:00 Reden: Ik keur dit document goed Getekend door: Wim Van Hoydonck (Signature) Getekend op: 2021-02-11 15:58:03 +01:00 Reden: Ik keur dit document goed <i>Kristof Verelst</i> <i>Jeroen Verwilligen</i> <i>Wim Van Hoydonck</i>
Project leader:	Vercruysse, J.	Getekend door: Jeroen Vercruysse (Signature) Getekend op: 2021-03-09 17:15:03 +01:00 Reden: Ik keur dit document goed <i>Jeroen Vercruysse</i>

Approval

Head of division:	Mostaert, F.	Getekend door: Frank Mostaert (Signature) Getekend op: 2021-02-11 16:13:40 +01:00 Reden: Ik keur dit document goed <i>Frank Mostaert</i>
-------------------	--------------	--



Abstract

This work investigates the discharge capacity of a side conveyance structure located at lock No. 3 in the Dessel-Schoten canal. A first level of analysis involves the use of classical semi-empirical techniques available from hydraulics. A second level of analysis involves the use of Computational Fluid Dynamics (CFD), in an effort to reduce the uncertainties associated with the use of the aforementioned techniques. It is shown that, although in principle the flow present in the side drainage falls into one of the categories in which the proposed techniques are valid, discrepancies with the results obtained numerically suggest that the underlying assumptions in the technique need to be revised when used for cases with similar characteristics. This work also focus in reducing the impact of side flows for navigation through locks, associated with the operation of the side drainage structure. Different alternatives for safe navigation are proposed, and studied using CFD. In particular, four scenarios are studied, of which three comprise different improvements to the original structure. One of the proposals is found to have promising features, whilst being relatively simple.

Contents

Abstract	III
List of Figures	VI
Nomenclature	VII
1 Introduction	1
2 Geometry of the outlet structure as-built.....	4
3 Analysis of discharge through the drainage structure.	6
3.1 Governing equations of the flow	6
3.1.1 Full three-dimensional approach: Volume-of-Fluid (VoF).....	6
3.1.2 One-dimensional approach: Momentum and Energy (Bernoulli) equations	6
3.2 Verification of CFD model and 1-D calculations.....	7
3.2.1 Validation of CFD methodology	7
3.2.2 Analysis of the drainage structure: constrained inlet	8
3.2.3 Semi-empirical approach: discharge formulas and first principles	8
3.2.4 Previous drainage methodology: syphoning	10
3.2.5 CFD Analysis.....	10
3.2.6 Summary	11
4 Design of the outlet structure	15
4.1 Post-processing: Data visualization, Hydraulics through the culvert, and Navigability through the locks.....	15
4.1.1 Precedents: basis for post-processing.....	15
4.1.2 Data visualization: common features	15
4.1.3 Frame of reference: From where are we observing?.....	16
4.1.4 Assessing Navigability	17
4.1.5 Assessing culvert's performance: Backwater calculations	17
4.2 Numerical simulations without outlet structure.....	20
4.2.1 Open-channel hydraulics: partially drowned flow	20
4.2.2 Navigability Assessment	21
4.3 Numerical simulation of design alternatives	22
4.3.1 First alternative: side channel with sill	23
4.3.2 Second alternative: side channel with stepped slope	26
5 Discussion.....	31
References	33
A1 On a different alternative	A1
A2 Technical Drawings.....	A4

List of Figures

Figure 1	Schematic top view of the Dessel-Schoten Lock system.	1
Figure 2	View of the drainage structure from Lock No. 3 in Dessel-Schoten.	2
Figure 3	Determination of discharge for a highway circular Culvert, following Bodhaine (1968)	3
Figure 4	Geometry of existing lock structure and main channel.	4
Figure 5	Cross-section and bathymetry of the main channel.	5
Figure 6	Test case used for the 'validation' of the CFD results.	7
Figure 7	Results obtained from OpenFOAM for the archetype case.	12
Figure 8	Flow analysis across the inspection boxes.	12
Figure 9	CFD results obtained for the drainage structure proposed for the lock	13
Figure 10	An inspection chamber with a submerged intake.	14
Figure 11	Standard streamwise sections for post-processing.	16
Figure 12	Illustration of the results for the structure currently in place.	20
Figure 13	Free surface inside the culvert.	21
Figure 14	Streamwise section colored by transversal (plane normal) velocities.	22
Figure 15	Draft-averaged transversal velocities along the streamwise sections.	23
Figure 16	Geometry and dimensions of first alternative.	24
Figure 17	Simulation results for first alternative.	25
Figure 18	Draft-averaged transversal velocities along the streamwise sections.	26
Figure 19	Geometry and dimensions of second alternative.	27
Figure 20	Draft-averaged transversal velocities along the streamwise sections.	28
Figure 21	Volume streamlines of the mean flow around the confluence.	29
Figure 22	Simulation results for the second alternative.	30
Figure 23	Geometry and dimensions of third alternative.	A2
Figure 24	Simulation results for third alternative.	A3

Nomenclature

Abbreviations

CAD	Computer-Aided Design
CFD	Computational Fluid Dynamics
dVW	de Vlaamse Waterweg nv
ECMT	European Conference of Ministries of Transport
FHR	Flanders Hydraulics Research
FOSS	Free and Open-Source Software
FVM	Finite Volume Method
LOA	Length Overall of a vessel
OpenFOAM	Open-Source Field Operation And Manipulation library
RAS	Reynolds-Averaged Simulations
VoF	Volume of Fluid method

Latin symbols

b	Width	m
D	Diameter	m
d_m	Effective rock size	m
F	Force	N
f	Hagen-Poiseuille friction coefficient	—
g	Gravitational acceleration	m/s ²
K	Hydraulic Conveyance coefficient	m ³ /s
L	Length	m
$\dot{m}$	Momentum flux	N/s
n	Glauckler-Strickler-Manning coefficient	m ^{1/3} /s
p	Instantaneous dynamic pressure	Pa
Re	Reynolds number	—
U	Mean velocity	m/s
u	Instantaneous velocity	m/s
y	Height	m
y_c	Critical depth	m

Greek symbols

Δh	Head loss	m
α	Indicator function	—
ζ	Local loss coefficient	—
ρ	Density	kg/m ³
τ^r	Residual stress	Pa
ϵ	Pipe roughness	mm

1 Introduction

The Dessel-Schoten canal offsprings in Dessel where the Bocholt-Herentals canal and the Dessel-Kwaadmechelen canal meet. Downstream, at Schoten, the canal discharges into the Albert Canal. The canal is composed of 10 locks: the first lock is situated in Rijkevorsel, the other 9 are situated in Brecht, Sint-Job-in't-Goor and Schoten¹. With the exception of the last lock in Schoten, the other 9 locks are identical. These locks have a useful length of 50 m and a useful width of 7.0 m, making the channel accessible for European Conference of Ministries of Transport (ECMT)-class II ships (600 tonnes). Most of the channel's locks do not have side drainage facilities. The hydraulic head along the locks is approximately 2.4 m with the exception of the last lock for which the head is 5.7 m. This work will focus on Lock No. 3, located in Brecht (see Figure 1).



Figure 1 – Schematic top view of the Dessel-Schoten Lock system. Source: binnenvaart.be

Notice that in order to guarantee general mass balance in case of leakage losses, regulate the water level on each of the reaches, and compensate the extra mass loss caused by the higher head in the last lock, drainage is needed. With the exception of the last lock, draining is done by operating the levelling drains at the doors or by means of overtopping the gates. In the future the locks on the canal will be operated remotely. This requires that a navigation lock doesn't function as a drainage structure. Therefore a separate drainage culvert needs to be foreseen alongside each lock.

The proposition of drainage and side discharge structures is commonplace in the design of locks, fish passages, storm basins and, related hydraulic infrastructure. These drainage structures can come in the form of weirs, side culverts, or overtopping sections within the structure. In the case of lock systems in canals such flow diversions help regulate the mass loss across the system caused by the operation of each independent lock during the passage of a vessel. Sometimes, depending of whether the canal has tributaries pouring into it, side weirs are used for such purpose. Such discharges are located laterally, along the exit of the lock, and sometimes put in pairs in an attempt to cancel the cross forces of the incoming jets in the mid axis of the channel. Particularly, in Flanders, culverts with submerged outlet are the standard option.

Here, a single culvert is proposed to go on either side of the lock with a free outfall. The entrance to the pipe is composed of a pair of inspection boxes: one connecting the pipe in a perpendicular bend to the second box, connecting to the upstream canal, through a slit near the bottom. Strictly speaking, the box facing the upstream canal is not an *inspection* box but an *intake* with similar dimensions to the box connecting to the pipe. For the sake of simplicity, both accessories will be referred to as inspection boxes.

¹De Vlaamse Waterweg nv - Afdeling Regio Oost is the waterway administrator in charge of the canal Dessel-Schoten



Figure 2 – View of the drainage structure from Lock No. 3 in Dessel-Schoten.

At Lock No 3 of the Dessel-Schoten canal, the waterway administrator already installed a drainage structure consisting of a single pipe next to the lock (see Figure 2). This structure however was put as a provision, hence its dimensioning was made in situ and its hydraulic performance not analysed. However, Flanders Hydraulics Research (FHR) was asked by de Vlaamse Waterweg nv (dVW)-Afdeling techniek² to perform a hydraulic study on Lock's No. 3 drainage structure and to propose the necessary modifications for the safe operation of the lock due to the flow through the side drainage; the proposed modifications to the current installation may serve as a template for the side drainages to be constructed.

Such configuration poses challenges in terms of the assumptions one needs to make, or accept, when using semi-empirical formulae for determining the discharge. One salient semi-empirical work is that of Bodhaine (1968) where, according to some criteria, the discharge across a culvert of circular section may be determined as shown in Figure 3. Notice that this work assumes the culvert has no geometrical constraints at the inlet, nor in the outlet, which is a reasonable assumption for drainage structures in highway embankments and cofferdams. However, the discharge capacity of a channel is conditioned almost entirely by the *end* conditions of the system, and on a lesser extent to head losses across the system itself. The Author believes that the semi-empirical methodologies that are common for the design of pipe culverts fall short in accounting for changes in momentum caused by geometrical constraints at the inlet of the pipe, thus leading to an overestimation of the discharge when using semi-empirical techniques. The problem with the aforementioned models is in the difficulty of determining which assumptions are valid and which are not, particularly when determining the *hydraulic control*. Here, a deterministic approach using tools of fluid mechanics, and CFD, will be pursued in order to establish the discharge. A three-dimensional simulation may offer insight on the nature of the flow around the inspection boxes and pipe.

²Contact: Eddy Vervoert, P. E.

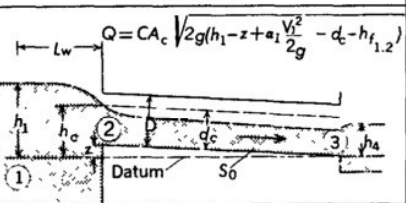
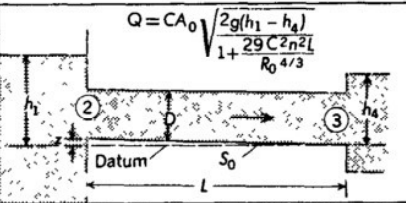
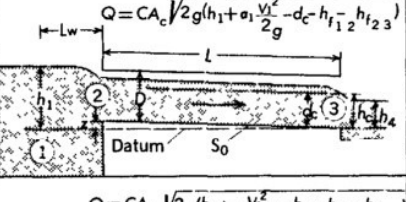
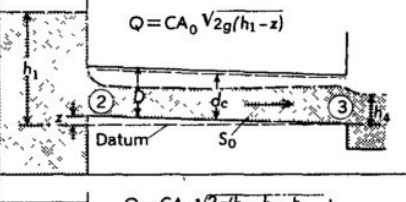
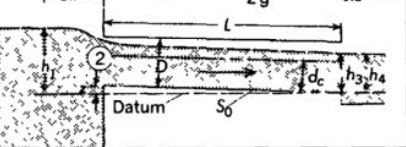
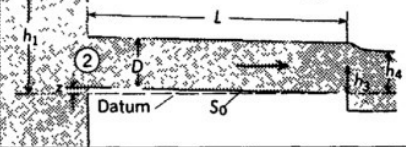
TYPE	EXAMPLE	TYPE	EXAMPLE
1 CRITICAL DEPTH AT INLET $\frac{h_1 - z}{D} < 1.5$ $h_4/h_c < 1.0$ $S_0 > S_c$		4 SUBMERGED OUTLET $\frac{h_1 - z}{D} > 1.0$ $h_4/D > 1.0$	
2 CRITICAL DEPTH AT OUTLET $\frac{h_1 - z}{D} < 1.5$ $h_4/h_c < 1.0$ $S_0 < S_c$		5 RAPID FLOW AT INLET $\frac{h_1 - z}{D} > 1.5$ $h_4/D \approx 1.0$	
3 TRANQUIL FLOW THROUGHOUT $\frac{h_1 - z}{D} < 1.5$ $h_4/D \approx 1.0$ $h_4/h_c > 1.0$		6 FULL FLOW FREE OUTFALL $\frac{h_1 - z}{D} \approx 1.5$ $h_4/D \approx 1.0$	

Figure 3 – Determination of discharge for a highway circular Culvert, following Bodhaine (1968)

This first part of the study starts with a brief description of the mathematics involved in the numerical simulation, where details on the 1-D & 3-D governing equations for free-surface flow used are given. Then, a study case of an archetype flow (pipe flow between tanks) is used as a verification for the CFD code, comparing it with results from 1-D calculations. Afterwards, a one-dimensional analysis of the inlet before the pipe culvert in the lock is given, where different assumptions to those made previously are defined. Finally, the first part closes with an analysis of the results obtained using CFD for the discharge on the drainage pipe for Lock No. 3. Notice that once the discharge through the culvert is determined, further analysis downstream of the culvert will not require modelling the region upstream from the culvert's outlet: setting a constant discharge across the culvert outlet will be sufficient.

The drainage structure built in Lock No. 3, as-is, lacks an appropriate transition to the main channel downstream. Thus, the second part of this work deals with the proposal and study of a side channel at the outlet of the culvert. This structure is thought to reduce transversal flows that may hamper navigation of vessels entering or exiting the downstream lock head, while operating the drainage structures. An original proposal, and a subsequent improvement are presented and modelled. An analysis on the transversal velocities exiting the dissipation structure into the main channel is made, and used as an indicator of the navigability through the lock system. A description of the proposals will be made, along with their respective CFD results. A relatively different proposal, with its respective hydraulic performance, is provided in the Appendix A.

This report is split in two parts: (1) one dealing with the calculation of the discharge passing through the culvert, and (2) a second part dealing with the design of an outlet structure connecting the side discharge with the main channel. The reason for this division is discursive: one part pertains the analysis of certain conditions over the existing structure, while the other part is *propositive*, that is, an outlet structure is proposed and new proposals are drawn from the results of simulations obtained from the previous proposal. Notice that each part may be read independently from the other, without loss of continuity.

2 Geometry of the outlet structure as-built

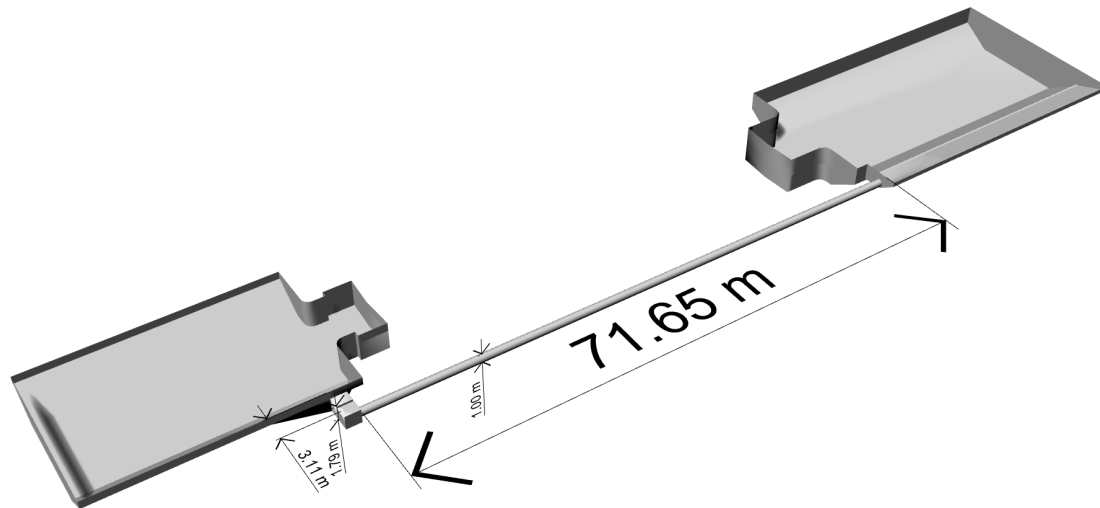
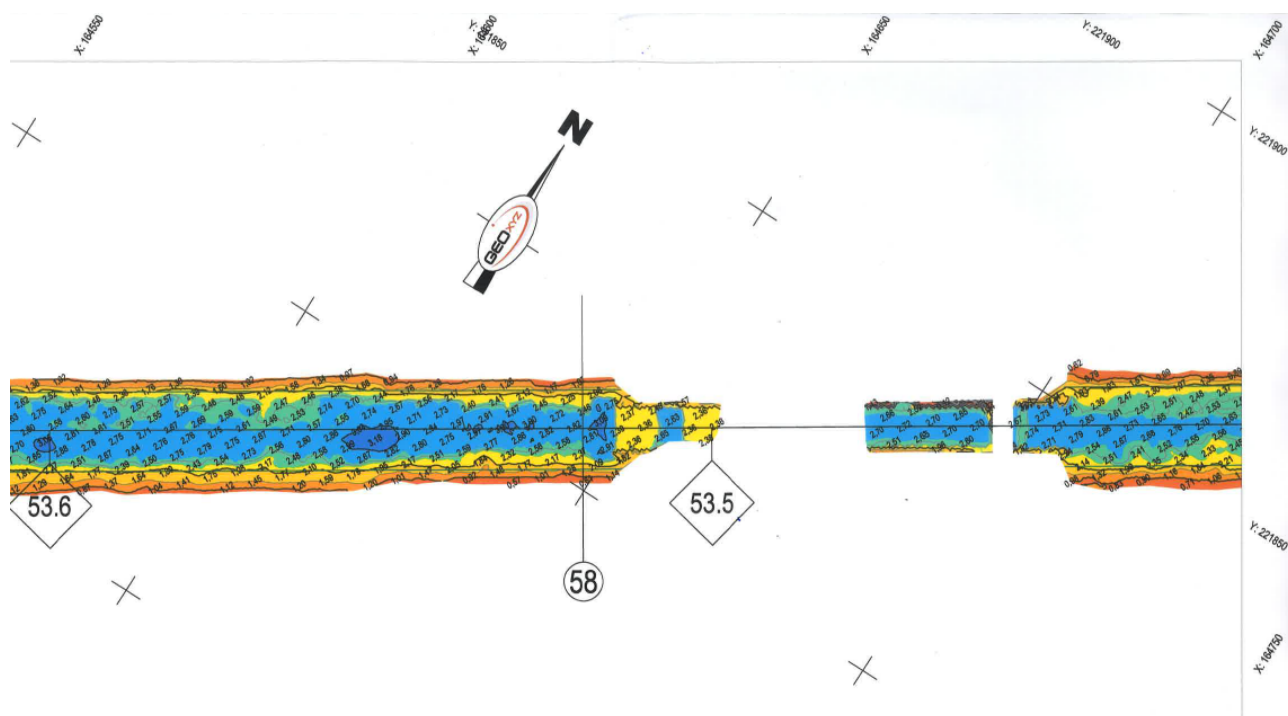


Figure 4 – Geometry of existing lock structure and main channel.

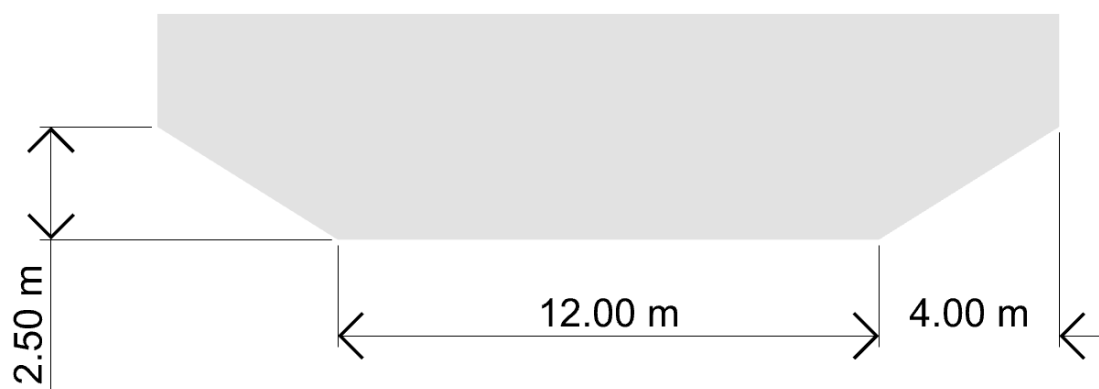
A first step into the numerical simulation of the hydraulic structure is the three-dimensional digitalization of the technical drawings provided by dVW. A 3D Computer-Aided Design (CAD) drawing of the structure as-built is presented in Figure 4. The digitalization of some features required good judgement and, to some degree, simplifications, since the data presented here and the drawings presented in Appendix A2 may need completion. It is very common for this kind of projects to have differences between the structures at post-construction and the one proposed at the design phase. This study will take special care on the measurements involving the inlet and outlet of the culvert, and the dimensions of the inspection boxes. Since one of the main objectives of the present work is the determination of discharge passing across the structure, it is of prime importance to determine the slope of the pipe *as-built*. For this, the following analysis was made:

- After discussions with dVW, it was decided that the water depth in either channel is 2.5 m. After observing the bathymetry in Figure 5, the cross section of the channels upstream and downstream of the lock may be assumed as trapezoidal (see aforementioned Figure for details). By using the reference height of the water level upstream and downstream from the lock, one can construct the geometry of the channels.
- The reference height for the water level upstream from the lock is 26.196 mTAW and the height of the bottom of the inspection chamber is 24.663 mTAW. If we add to the latter the height of the inspection chambers, which is 1.79 m, we find a reference height of 26.453 m, which is above the mean water level. Notice that elevations of the inspection boxes exist (see Appendix A2) but these are too different from one another, hence they were not considered reliable. The information about the inspection boxes were extracted from the as-built technical drawings.
- The reference height for the water level downstream of the lock is 23.524 mTAW.
- Following the technical drawings, the location of the culvert's bottom at the inlet is 15 cm above the inspection chambers' floor, leading to a reference height of 24.813 mTAW. The reference height of the culvert at the outlet is 24.782 mTAW, leading to a drop of 3.1 cm between the inlet and the outlet.

In the preparation of the simulations, it was assumed that the zero-height was set on the bottom of the downstream channel. However, when necessary, such reference may be placed elsewhere.



(a) Bathymetry survey made on 28/05/2013.



(b) Reference cross-section.

Figure 5 – Cross-section and bathymetry of the main channel. Information provided by dVW.

3 Analysis of discharge through the drainage structure.

3.1 Governing equations of the flow

This section will give a brief description of the mathematics used to model two-phase flows, either three-dimensionally or just in one dimension. This section is complementary to the semi-empirical method mentioned in the Introduction and will expand on it based on *first principles*. This section may be skipped, without loss of continuity.

3.1.1 Full three-dimensional approach: Volume-of-Fluid (VoF)

The governing equations, written in indicial form ($i, j = 1, 2, 3$), of a two-phase flow modelled via Volume of Fluid method (VoF) in a cartesian frame of reference are

$$\partial_i u_i = 0, \quad (1)$$

$$\partial_t (\rho u_i) + \partial_j (\rho u_j u_i) = -\partial_i p + \partial_j (\tau_{ij} + \tau_{ij}^r) + \rho g_i, \quad (2)$$

where u , g , p and ρ are the flow velocity, gravity, dynamic pressure (no static contribution), and density, respectively. Notice that surface tension is ignored in this formulation. Since the comprising phases are air and water, the internal shear stresses can be modelled as Newtonian:

$$\tau_{ij} = \nu (\partial_j u_i + \partial_i u_j), \quad (3)$$

and the residual stress τ^r can be modelled using any turbulence closure model (Wilcox *et al.*, 1998). Also note that the density of the non-miscible fractions can be defined as follows:

$$\rho = \alpha \rho_{\text{water}} + (1 - \alpha) \rho_{\text{air}}, \quad (4)$$

where α is an indicator function. Such indicator must be conserved at each fluid parcel, that is:

$$\partial_t \alpha + \partial_i (\alpha u_i) = 0. \quad (5)$$

Several methodologies exist for the numerical solution of the equations just presented. Here the Finite Volume Method (FVM) is the preferred method to use, and Open-Source Field Operation And Manipulation library (OpenFOAM) the software of choice for conducting the simulations. Please refer to Rusche (2003) for details on the implementation present in OpenFOAM.

3.1.2 One-dimensional approach: Momentum and Energy (Bernoulli) equations

In addition to the semi-empirical methodology mentioned in the Introduction, the analysis of the discharge across the inspection boxes may be verified using the Bernoulli Equation and Newton's second law around an arbitrary control volume:

$$y_1 + \frac{U_1^2}{2g} = y_2 + \frac{U_2^2}{2g} + \Delta h, \quad (6)$$

$$\sum_f \mathbf{F} = \sum_f \dot{\mathbf{m}}, \quad (7)$$

where the sub-indices 1 and 2 indicate the endpoints along a streamline, U the *mean* streamwise velocity, y the water depth, F a vector force, and $\dot{m}$ impulse. Note that the summation of forces/impulse occur along the faces of the control volume chosen. The head losses due to channel form changes and turbulence are condensed in Δh .

3.2 Verification of CFD model and 1-D calculations

This particular section will delve into the validation of the discharge obtained both by CFD and semi-empirical methods. Although generally semi-empirical methods for the determination of discharge in structures under steady-state conditions is rather accurate, the particularities present in the drainage structure of Lock No. 3 demand a further verification. Here, a validation of the numerical tools proposed is made; then, this section closes with a study of the discharge in the drainage structure using semi-empirical methods and CFD.

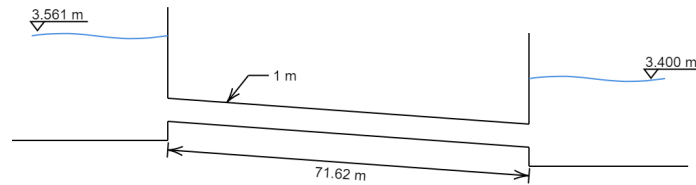


Figure 6 – Test case used for the ‘validation’ of the CFD results. Note that the dimensions of the pipe are similar to those for the lock’s drainage structure.

3.2.1 Validation of CFD methodology

As a validation for the CFD-methodology being used, a simple archetype flow is chosen with respect to one-dimensional solutions. Such case is pressurized pipe flow of water between two tanks, as depicted in Figure 6. The roughness of the pipe will be set to 0 mm, which assumes hydraulically smooth turbulence regime. The net loss coefficients due to the entrance and the exit of the pipe will be considered equal to 1.5.

A very simple use of Equation 6 shows that for the pipe with flowing water the energy is equal to

$$\Delta h = \left(\zeta + f \frac{L}{D} \right) \frac{U^2}{2g},$$

$$0.161 \text{ m} = \left(1.5 + f \frac{71.62}{1} \right) \frac{U^2}{2g},$$

where f , L , D , and ζ are the friction factor, pipe length, diameter, and local contraction losses, respectively. The friction factor can be determined using the Colebrook-White formula:

$$\frac{1}{\sqrt{f}} = -0.86 \log \left(\frac{\epsilon}{3.7 \cdot D} + \frac{2.51}{\text{Re} \cdot \sqrt{f}} \right), \quad (8)$$

$$\frac{1}{\sqrt{f}} = -0.86 \log \left(\frac{2.51 \cdot 1 \times 10^{-6}}{U \cdot 1 \cdot \sqrt{f}} \right),$$

where Re is the *Reynolds number*, and ϵ ($= 0$ mm) is the roughness of the pipe. By solving the previous system of equations using a fixed-point iteration method one obtains that:

$$\begin{aligned} U &= 1.159 \text{ m/s}, \\ f &= 0.01154, \end{aligned}$$

hence the discharge is equal to $Q = 0.910 \text{ m}^3/\text{s}$.

Results obtained using CFD are shown in Figure 7. The discharge calculated there is equal to $Q = 0.650 \text{ m}^3/\text{s}$, determined by integrating the two-dimensional velocity profile on a cross section of the pipe. Notice that the discharge given by the CFD model is *somewhat* lower compared to the results obtained using 1-D calculations. The reason for these differences is that the discharge is very sensitive to small changes in the friction factor which, given the assumptions of 1-D models, are estimations on a first level of analysis. For instance, taking the average velocity given by the discharge obtained using CFD, one gets $f_{\text{CFD}} = 0.012032$ which only differs 4.1% compared to the 1-D results. Furthermore, the local losses used in this case are only assumptions, using values common in engineering practice. In general, both components of energy loss contribute in the uncertainties inherent in the determination of discharge.

One could refine the one-dimensional calculations by extracting the median heights of the reservoirs extracted from the CFD. By measuring the relative median heights, or $\Delta h = 0.15923$ m, between the reservoirs shown in Figure 7(a), and setting $\zeta = 3.65$, the discharge predicted by CFD is obtained by solving equations 6 and 8.

3.2.2 Analysis of the drainage structure: constrained inlet

The necessary geometrical features of the drainage structure are obtained from the 3D solid presented in Figure 4. The entrance to the pipe is made of two inspection boxes interconnected through a throat of dimensions 1.4×0.7 m. The geometrical details of the inspection boxes were extracted from the technical drawings presented in Appendix A2. These boxes connect the channel and the pipe perpendicularly, constraining the flow at the inlet to the changes in momentum caused by such an abrupt change of direction. Additionally, the bathymetry shown suggests that the channel is of a trapezoidal shape. For the sake of simplicity, the cross sections of the canal upstream and downstream of the lock are assumed prismatic and of trapezoidal section.

3.2.3 Semi-empirical approach: discharge formulas and first principles

First, a semi-empirical approach to the determination of the discharge is followed. The classical method presented in Bodhaine (1968) is employed. This method proposes different approaches into obtaining the discharge of highway crossings and culverts, depending on the type of flow that it may occur within the structure. The first criterion needed to determine the type of flow, the so-called relative submergence, depends on the height of the water column above the culvert's floor at the inlet, $h_1 - z$, and the diameter, D , of the conduit. Its calculation gives:

$$\frac{h_1 - z}{D} = \frac{1.383 - .031}{1} = 1.352 < 1.5,$$

which corresponds to critical flow at the outlet (type II). Notice that the notation of the sub-indices indicate the different cross sections shown in every flow type in Figure 3. Following the Figure, the discharge may be

calculated as

$$Q = C A_c \sqrt{2g \left(h_1 + \frac{\alpha_1 V_1^2}{2g} - \Delta h_{f_{1-2}} - \Delta h_{f_{2-3}} \right)}, \quad (9)$$

$$\Delta h_{f_{1-2}} = \zeta_{1-2} \frac{Q^2}{2g A_{1-2}}, \quad (10)$$

$$\Delta h_{f_{2-3}} = L(Q^2/K_2 K_3), \quad (11)$$

$$K = (1/n) R^{2/3} A. \quad (12)$$

Notice that many of the variables have been presented already. In particular K , the hydraulic conveyance, is a measure of the transport *potential* of a channel under normal flow conditions. This expression is better represented using Manning's equation, where n is a measure of energy loss. Notice that the total head in the entry section can be simplified as

$$h_1 + \frac{\alpha_1 V_1^2}{2g} \approx H_3 - H_2 = 1.344 \text{ m}, \quad (13)$$

where H is the vertical datum at a section. Remember that the subindices 2 and 3 correspond to the inlet section and the outlet section of the culvert, respectively. This approximation follows from the proposition that the entrance section can be considered 'ponded'. By corollary, the calculation of the hydraulic conveyance at the inlet of the culvert, K_2 , will be done for a fully wet section. Notice that this assumption goes in *contradiction* with the type of flow proposed (type II) as basis for the analysis thus, strictly speaking, rendering any subsequent calculation *incorrect*. However it will be later shown that such assumption is correct, in light of the results that will be obtained using CFD.

Continuing with the calculations, the Manning's coefficient will be considered equal to $n = 0.015$. The contraction coefficient for an orifice can be considered equal to $C = 0.61$, and the loss coefficient at section 1-2 equal to $\zeta_{1-2} = 0.5$. By using the following approximation for the calculation of y_c in circular pipes

$$y_c = \frac{1.01}{D^{0.26}} \left(\frac{Q^2}{g} \right)^{0.25}, \quad (14)$$

one can determine the discharge to be equal to $Q = 1.402 \text{ m}^3/\text{s}$. A verification on whether the flow is indeed drowned in the inlet section of the culvert is in place.

This verification involves the one-dimensional study of the flow across the inspection boxes, as shown in Figure 8. For the sake of simplicity, the width of the inspection boxes are considered equal to the width, b , of the throat, that is 1.4 m. The height of the opening is 0.7 m and the water depth at the entrance of the system is equal to $y_0 = 1.533 \text{ m}$. Conservation of energy dictates that:

$$y_0 + \frac{Q^2}{2g(b \cdot y_0)^2} = y_2 + \frac{Q^2}{2g(\pi \cdot D^2/4)^2} + \Delta h \quad (15)$$

$$y_0 + \frac{Q^2}{2g(0.7 \cdot y_0)^2} = y_2 + \frac{Q^2}{2g(\pi/4)^2} + \Delta h. \quad (16)$$

By applying Newton's second law on the control volume depicted in Figure 8, one gets the following relation:

$$\rho Q V_1 = -\frac{1}{2} \rho g b y_2^2 + \frac{1}{2} \rho g b y_1^2 \quad (17)$$

$$\frac{\rho Q^2}{1.4 \cdot 0.7} = -0.7 \cdot (\rho g) y_2^2 + 0.7 \cdot (\rho g) (y_0 - \Delta h - \varepsilon^2)^2, \quad (18)$$

where on the right hand side one has the momentum fluxes at the intake from the channel, balanced by the pressure forces from the intake section and at a section just downstream from the throat. The term ε is a small number ($\ll 1$), used to avoid getting a trivial solution for Equations 16-18.

If one assumes a head drop Δh^3 of 13 cm to happen just after the contraction, the discharge and piezometric head at the pipe becomes

$$Q = 1.257 \text{ m}^3/\text{s},$$

$$y_2 = 1.2945 \text{ m},$$

showing that the flow at the entrance of the culvert is effectively drowned .

Note that this level of analysis gives a discharge somewhat lower from the one calculated previously. For the sake of consistency, it is important to enumerate the assumptions made for the study of this particular case:

- No friction: this particular assumption seems reasonable for the dimensions considered in this case.
- The contraction of the flow only happens vertically: by assuming the width to be equal across the inspection boxes the lateral contraction of the flow is neglected. On one end, the definition of the local losses is simple; on the other end, the head drop across the throat will be underestimated.
- The pipe is **drowned**: this assumption may go in contrast with the submergence ratio criterion which dictated that for this particular case the pipe was *not* going to be drowned.
- The pressure distribution around the submerged gate are varying uniformly. In reality, near the opening, pressure distributions become concave (as seen from the wall) as local accelerations become higher. This results in a lesser force acting on these walls.

These assumption are, however, also present in the tools proposed by Bodhaine (1968). This is an indication that, in light of the results obtained with CFD, the hydraulic control is indeed the inlet of the culvert. The lower discharge obtained with the latter methodology reinforces this fact.

3.2.4 Previous drainage methodology: syphoning

Incidentally, before the construction of the side drainage in Lock No. 3, the dVW conducted drainage operations by *syphoning* water across the lock. In particular a pipe similar to the one currently in use, but of 600 mm diameter, was placed at each lock's ends and used as a syphon. The height difference between the syphon's ends was determined to be 2.67 m. With this information, we can determine the syphon's discharge just by solving Equation 9 and 12 iteratively. Note that the hydraulic conveyance of the syphon both at the inlet and outlet are for a fully wet pipe section, the available head is equal to the height difference between the syphon's ends, and the length of the syphon is 75 m. The discharge obtained is $Q = 0.793 \text{ m}^3/\text{s}$. The client expects that any new drainage structure should provide at least the discharge provided by the syphon.

3.2.5 CFD Analysis

As it can be seen from the previous discussion two methodologies following conflicting assumptions give similar discharges. The outcome of the CFD simulations might offer a *consensus* on the approach that may be followed for this type of cases. For instance, one of the assumptions was that the hydraulic control is given by the 90-degree bend between the inspection box and the pipe culvert. Instead, the hydraulic control is exactly at the culvert inlet. However, one should not discard the necessity of performing in-situ measurements given the uncertainties that are common in the construction of this type of projects.

³This value is extracted according to the CFD results shown in the next section

Results obtained from the numerical simulation of the drainage structure are shown in Figure 9. Note that there is a drop in water level across the inspection boxes (13 cm), confirming the assumption of submergence at the entrance of the pipe. Nonetheless, notice that air enters intermittently through the pipe inlet; such type of cases are best avoided when preparing designs using the classical tools of hydraulics. In this case, given the particular construction at the culvert's inlet, it was difficult to determine empirically whether partial aeration was occurring.

Also, it is important to mention that the flow in the pipe *changes* regime: from drowned to free-surface flow. From the two dimensional velocity profile obtained from the cross section shown in Figure 9(c) a discharge of $Q = 1.449 \text{ m}^3/\text{s}$ is obtained, similar to the results calculated by the one-dimensional approach. Such approach however, needed further assumptions that were made only in light of the results obtained with CFD calculations.

3.2.6 Summary

There are two main features of the current flow that are acting as hydraulic controls along the drainage structure, according to the CFD results:

- (i) The inspection box inlet perpendicular to the stream. As seen previously, the abrupt changes in direction of the flow entering the pipe transform part of the incoming kinetic energy into potential energy at the pipe's entrance. This phenomenon partly explains the reasons for having an intermittently submerged pipe, despite the submergence ratio being so low.
- (ii) The flow changes regime inside the pipe. Note that for the first tens of meters, the flow in the pipe is drowned. Afterwards the flow detaches from the crown of the pipe becoming a free-surface flow. The carrying capacity for free-surface flows in pipes is generally lower with respect to fully drowned pipes with low bed slopes. In this case the bed slope is equal to $S = 0.031/71.65 \times 100 = 0.043 \%$.

Note that none of the semi-empirical methodologies herein used have been thought, or calibrated, to be used when air pockets enter the culvert. Notice also that the CFD results for the validation case predicts reasonable discharges with respect to the 1-D calculations; thus one might expect a similar situation to happen in the case being studied, if more accurate 1-D techniques were available. In this sense, the Author suggests prudence in the interpretation of said results.

On the other hand, much of the uncertainties related to the hydraulic analysis of the structure are caused by the inlet structure attached to the culvert. Lateral intakes from rivers and channels to small hydro-power plants or aqueducts may serve as guide for the design of the intake. In principle, the intake should transition from the main channel *smoothly* and avoid local losses that may be difficult to determine. It is also highly desirable to have either a fully submerged or partially filled inlet. Another alternative is then to consider a single inspection chamber, with a fully submerged orifice as an intake from the main channel. A second chamber, fully drowned and close to the atmosphere, may be considered for the placement of either an emergency or control valve. A sketch for such intake is proposed in Figure 10. A simpler approach would be to take the current design for the inspection chambers and deepen both the pipe's inlet and the bottom of said chambers. Notice however that for the geometry in question, there is only 3 cm difference between the ends of the pipe.

Furthermore, since the CFD results show that the *hydraulic control* is at the outlet of the culvert, instead of being in the intake, these adjustments are not *strictly* necessary for the operation of the outlet structure. However, attention must be paid at the intake: air pockets entering the pipe may force the flow's discharge to be very unsteady, and increases the possibility of leakage across the pipe junctions.

Finally, note that the discharge for the outlet structure in Lock No. 3 provides a discharge higher than the one expected when water was being syphoned across the lock.

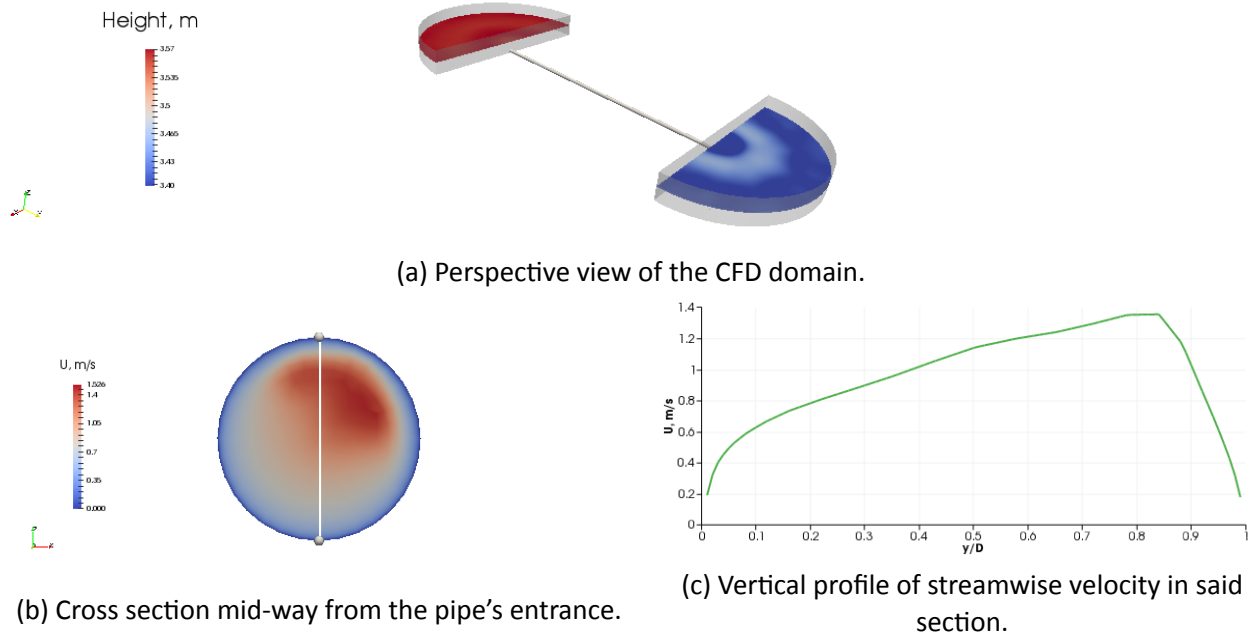


Figure 7 – Results obtained from OpenFOAM for the archetype case.

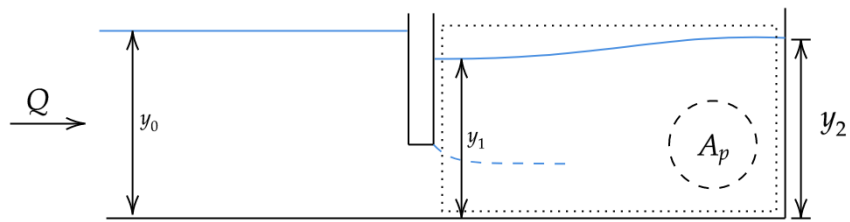


Figure 8 – Flow analysis across the inspection boxes.

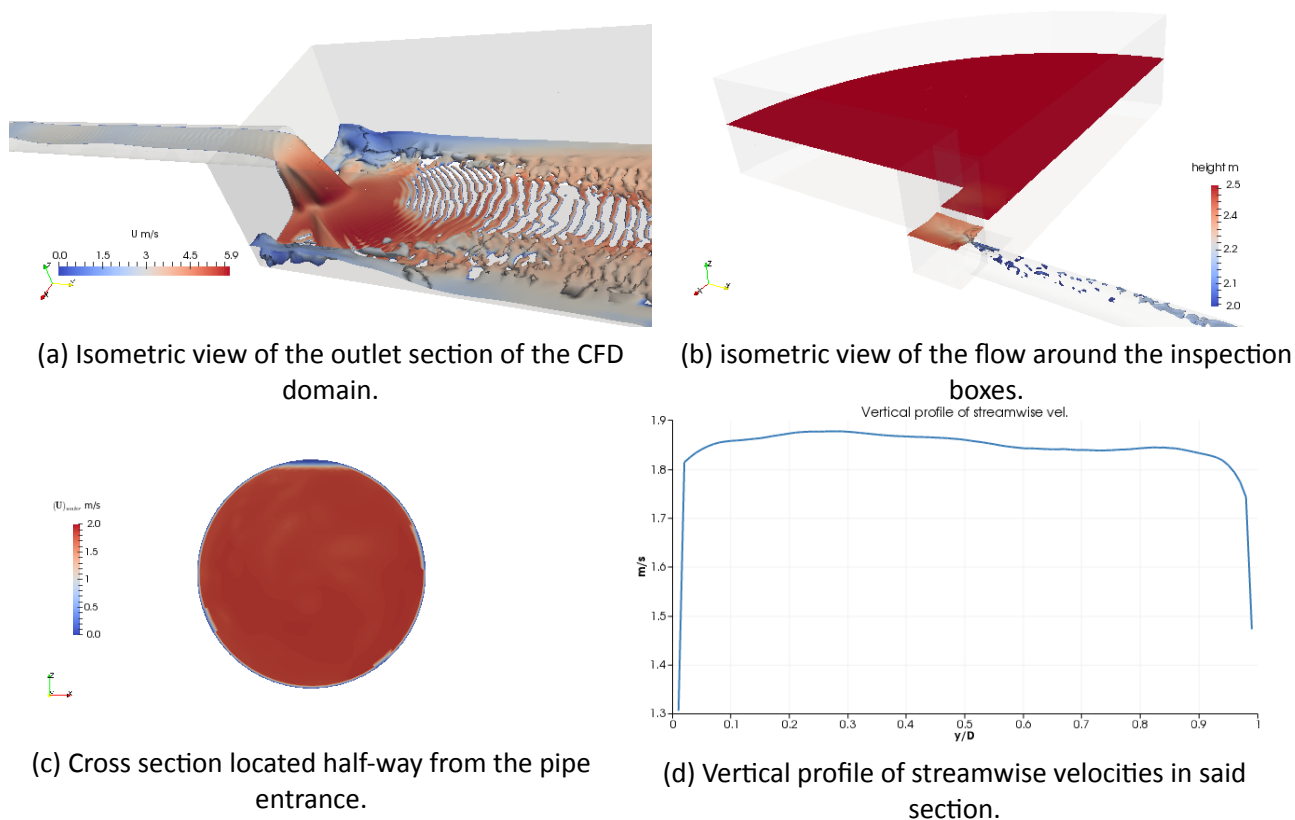


Figure 9 – CFD results obtained for the drainage structure proposed for the lock. The cross section of the pipe is taken half way of the pipe.

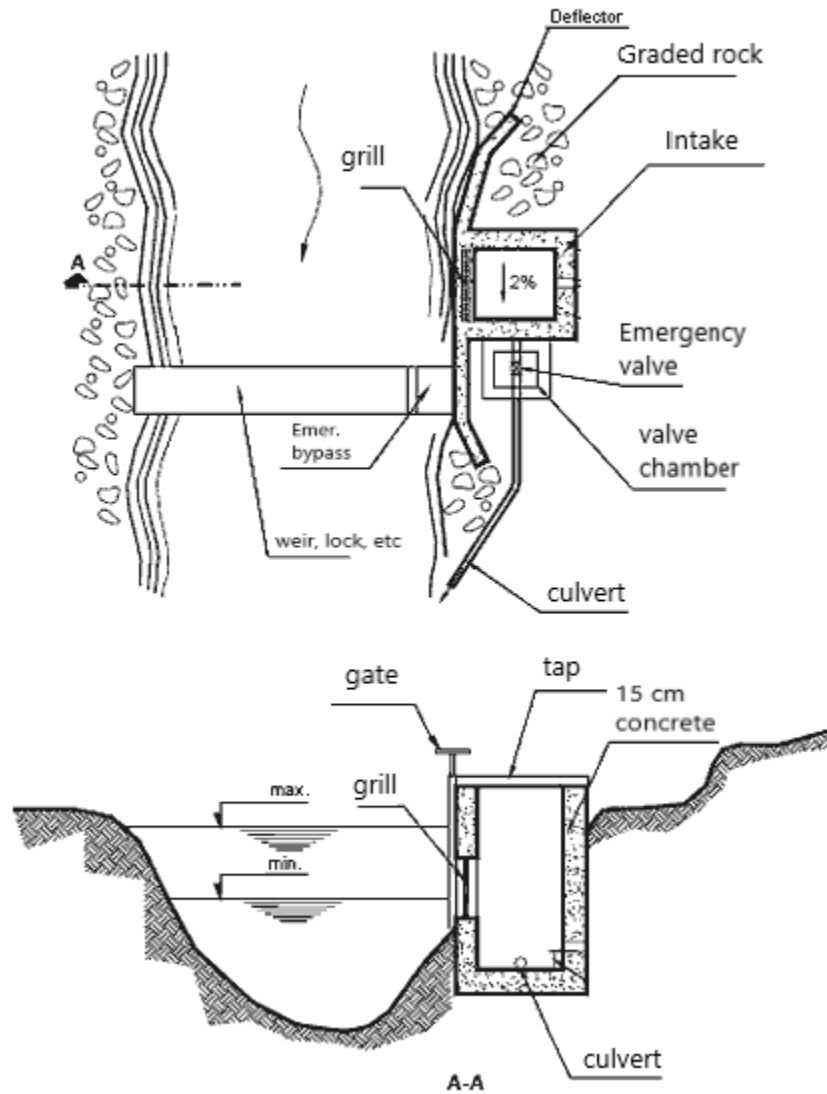


Figure 10 – An inspection chamber with a submerged intake. A single inspection box avoids the 13 cm head drop upstream of the culvert's entrance.

4 Design of the outlet structure

4.1 Post-processing: Data visualization, Hydraulics through the culvert, and Navigability through the locks

This chapter will describe some basic tools used for the analysis of the results that will be presented hereinafter. A first section will denote how some data from the CFD results will be presented. Then, the following section describes the tool used for assessing navigability, and a second section will present a simple backwater flow calculator used for the analysis of the flow through the side culvert.

4.1.1 Precedents: basis for post-processing

The discharge of the side culvert is needed as an input for the design of the outlet structure. Note that the operation of the discharge structure will be done to compensate for the mass losses occurring either naturally along the course of the canal (i.e. seepage, droughts) or by the operation of the lock system itself. In previous chapters it was shown that by having an unregulated⁴ side drainage, a discharge of approximately $1.5 \text{ m}^3/\text{s}$ was determined.

Being now the discharge a parameter of the simulation, a different research question may be addressed. A fundamental question in this context would be: Can the lock be operated (i.e. Can vessels enter or exit the lock safely) while discharging through the side drainage? Such mass imbalance is flushed laterally from the channel upstream of the lock onto the channel downstream. It is important that for the full working range of the bypass culvert ships can manoeuvre safely in- and out of the lock, despite the disturbance caused by the side drainage.

This disturbance is characterized by currents appearing laterally within the navigation channel, forcing the vessels to steer when encountered with such currents. According to Koedijk *et al.* (2017) guidelines, transversal velocities greater than 0.3 m/s may force the skipper either to manoeuvre (steering) against the currents or to be guided through the channel by external agents (i.e. guide wall, tow trains). Note that such guidelines assume the passage of recreational vessels affected by lateral flows acting over at least half its Length Overall of a vessel (LOA). Given the types of vessels crossing the canal and their LOA, a lateral discharge producing lateral velocities no higher than 0.3 m/s is sought. The velocities delivered by the lateral discharge may be modulated, if needed, by introducing an outlet structure. This chapter considers the design of the outlet structure for the drainage structure.

4.1.2 Data visualization: common features

In order to avoid unnecessary repetition, here we will describe the location and fields being extracted for visualization, and how they are presented. The post-processing of the results has been made in a way to guarantee uniformity between the different simulations, regardless of the geometric differences between the cases.

In general, all the field data extracted (i.e. velocities, water level) are *time-averaged*. The averaging window is of 3 minutes and 20 seconds, which is at least double the time taken for a parcel of fluid exiting the pipe to arrive to the outlet of the domain, assuming an average velocity of 1 m/s .

⁴Not constrained, i.e. knife gate fully open.

As stated previously, the main interest in this research is determining whether the transversal velocities exiting the side culvert affect navigation of a vessel entering or exiting the lock. In that case, there are three locations worth studying:

- Along the axis of the lock. We assume that the keel (or bow) of the vessel will be aligned with the axis of the lock, hence being this feature the first affected by the side currents. This might not be the case for vessels for which their hull follows a box shape.
- Along the 4.16 m axis starboard from the lock's axis. This location corresponds to the lock's sidewall, indicating a theoretical limit along which velocities shouldn't surpass a certain value.
- Along the 6.00 m axis starboard from the lock's axis. This location coincides with the toe of the channel's side bank and represents a theoretical limit across which vessels cannot navigate, thus along which transversal velocities achieve their maxima for the purposes of the present analysis.

Since the transversal velocities are the main concern at the locations just described only the streamwise plane sections are extracted. These locations are sketched in Figure 11. For the sake of simplicity in the analysis, the origin of the coordinate system will be part of a vertical plane containing the area comprising the outlet of the culvert. Its x-axis will be parallel to the channel mid-axis with its origin at the bed of the channel. The vertical z-axis' direction opposes the direction of gravity, and the y-axis is set such to follow the right hand rule. As a consequence, all positive transversal velocities in the streamwise plane point towards the reader.

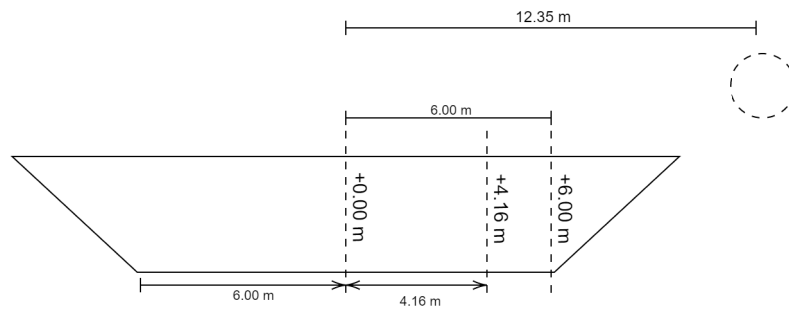


Figure 11 – Standard streamwise sections for post-processing.

4.1.3 Frame of reference: From where are we observing?

In order to ease the whole discussion when presenting the results it will be assumed that an observer is navigating the canal in a vessel from the west, towards the Albertkanaal. This implies that for said observer the channel westward from the lock will be also *upstream*, and viceversa. As a corollary, from the point of view of the vessel, the side channel and drainage are located on its starboard side. Notice that this convention directly connects with the point of view of an observer in hydraulics: one which looks in the direction where flow goes (or where, at least, potential energy decrease). In other words, when we refer to something upstream from the lock we are referring, for both observers, to the portion of the channel westward from the lock.

Note that this frame of reference is only *conventional*: vessels may cross the lock either from the west or from the east. The point is that for the purposes of this study, the actual location of the side drainage is indifferent for the analysis pertaining this work.

4.1.4 Assessing Navigability

The lateral velocity limit, as proposed, is rather *vague*. In an effort to make the analysis more concrete, a tool for the analysis of time-averaged transversal velocities is proposed. This tool seeks to summarize a given velocity field over a 2-D plane into a two-dimensional plot useful for assessing navigability.

To that end, a bivariate histogram is built where the spatial dimensions are binned as needed, and the value inside each bin is an appropriate lateral velocity. A natural definition of "appropriate" would be the average of the velocity values falling within each bin. This average, although valid, is not representative of the *forces* that may act across the chosen plane. A more appropriate measure would be to average as to preserve the net energy crossing the plane; that is, for a sequence of values (velocity field) in a normed space⁵ we can calculate its norm ('mean' velocity) as:

$$||u||_X = \frac{\sum_{X,Y}^N u_i |u_i|}{N}. \quad (19)$$

Notice that the resulting velocity at each bin in the histogram is

$$\bar{u} = \text{sgn}(|u|_X) \sqrt{|u|_X}. \quad (20)$$

Notice that by mapping the CFD calculations onto regular grids, the formulas just presented may be used directly.

4.1.5 Assessing culvert's performance: Backwater calculations

In order to accurately assess whether the assumptions on the *hydraulic controls* made in earlier sections is correct, free-surface profiles need to be determined. The free-surface may either be determined using one-dimensional techniques or extracted from the CFD calculations.

The aforementioned one-dimensional tool is implemented in the context of open-channel hydraulics. Although Free and Open-Source Software (FOSS) exist for one-dimensional backwater calculations, these codes integrate the constituting equations using one-sided Euler and upwind schemes. Such implementation allows for the study of open-channel flows in arbitrary geometries and cross sections, but may prove only approximate when working with regular cross sections, such as catenaries and pipe sections. The Direct Step Method (Chow, 1964), on the other hand, is exact for cross sections that can be described using algebraic equations such as the present culvert. A simple implementation for backwater calculations is presented in Listings 4.1. For the method to be exact, the normal depth must be calculated to machine accuracy, thus the algorithm for calculating the normal depth of various sections proposed is shown in Listings 4.2.

⁵ $\{u_i | x_i \mapsto u_i, x_i \in X, X \subset \mathbb{R}^2, ||x_i|| = c\}$, where c is a constant.

```

subroutine backwater_curve(flowprops , ystep , list )
  use M_linkedlist
  real , dimension(6) , intent(in)   :: flowprops ! Q, S, L, D, n, yinit
  real , intent(in)                  :: ystep
  type(linkedlist) , pointer         :: list

  real                                :: yend, ymid, dx, totL, yinit
  real                                :: S0, Se, H0, H1, Q, n, L, D, A, V

  Q    = flowprops(1)
  S0   = flowprops(2)
  L    = flowprops(3)
  D    = flowprops(4)
  n    = flowprops(5)
  yinit = flowprops(6)

  totL = L
  dx   = 0.0
  Se   = 0.0
  endless: do
    yend = yinit + ystep
    ymid = yend - 0.5*ystep
    A    = wettedArea(yinit ,D)
    V    = Q / A
    H0 = head (yinit , Q, D)

    call ll_insert(list , (/ totL , dx, yinit , H0, Se, A, V /))
    H1 = head (yend, Q, D)
    Se = normal_slope(Q, D, n, ymid)

    dx = ( H0 - H1 ) / ( Se - S0 )
    totL = totL + dx
    if ((totL <= 0) .or. (yend >= 0.95*D)) exit endless
    yinit = yend
  enddo endless

end subroutine backwater_curve

```

Listing 4.1 – Backwater flow algorithm.

```

function find_yn(Q, D, S, n)
  use autodiff
  real, intent(in) :: Q, D, S, n
  real :: find_yn

  type(autoderiv) :: xin, root
  logical :: found

  xin%v = rinit
  xin%dv = 1.0
  Q0 = Q
  D0 = D
  S0 = S
  n0 = n
  call find_root( gen_normal, xin, tolerance, root, found)

  call random_seed ()
  if(found) then
    do while ( root%v > 2.0*pi .or. root%v < 0)
      call random_number(xin%v)
      xin%v = xin%v*pi
      !print *, xin%v, root%v
      call find_root( gen_normal, xin, tolerance, root, found)
      if(.not.found) exit
    enddo
    find_yn = alphaToY(root%v,D)
  else
    find_yn = -1
  endif
contains
  function gen_normal(x)
    use autodiff
    type(autoderiv), intent(in) :: x
    type(autoderiv) :: gen_normal

    real :: a

    a = ((Q0*n0/sqrt(S0))**3.)*(D0/2.)**2.*(8./D0**2.))**5.

    gen_normal = a*x**2. - (x - sin(x))**5.
  end function gen_normal
end function find_yn

```

Listing 4.2 – Newton-Raphson algorithm using automatic differentiation, for the calculation of normal flow.

4.2 Numerical simulations without outlet structure

This chapter focuses on the numerical experiments conducted for the drainage structure in Lock No. 3 as-built in the Dessel-Schoten canal. The structure has been presented in Figure 4, and the calculation of its discharge studied in Section 3.2.5.

The information presented in the technical drawings in Appendix A2 are used to construct the computational domain. A more detailed depiction of the geometry and flow in the structure is shown in Figure 12.

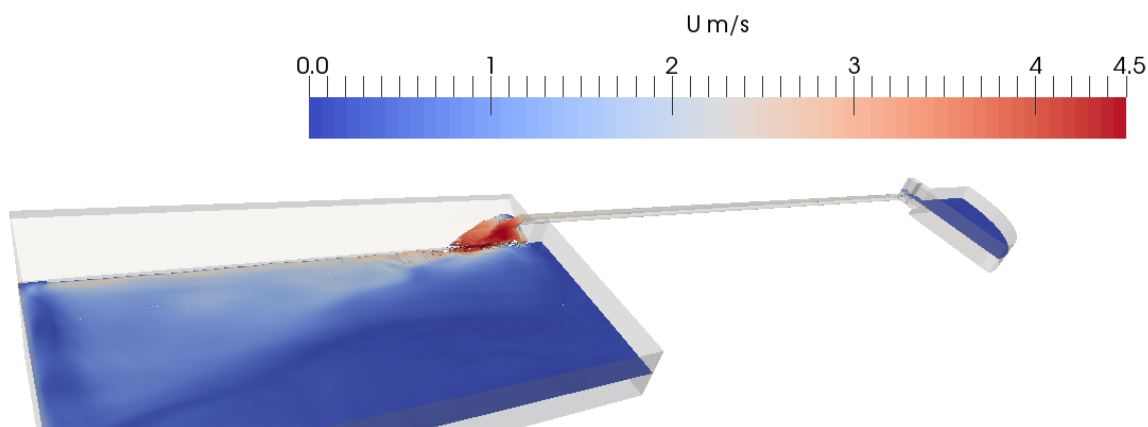


Figure 12 – Illustration of the results for the structure currently in place.

Here, the computational domain will encompass the channels upstream and downstream of the lock, as well as the side slope onto which the pipe culvert currently discharges. Furthermore, the initial conditions of the simulation, that is, the water levels upstream and downstream of the lock are assumed to be 2.5 m above the channels' bed. The corresponding discharge is set up to be $1.5 \text{ m}^3/\text{s}$ and no turbulence or, more precisely, fields modelling turbulence in a Reynolds-Averaged Simulations (RAS) context, is assumed entering the domain.

4.2.1 Open-channel hydraulics: partially drowned flow

As mentioned in previous chapters, and clearly seen in Figure 9(b), the flow is drowned inside the culvert for a short span. However, according to Bodhaine (1968), the relative submergence ratio predicts no drowning in the pipe.

In previous sections, it was shown that the hydraulic control was at the inlet section. This was shown to be the case by inspecting directly the flow at the inspection boxes, in the results obtained using CFD. A quantitative approach may extract the free-surface profiles from the numerical simulations, and then compare those with reduced-order models. The methodology of Bodhaine (1968) relies heavily on backwater calculations for the determination of the hydraulic conveyance at both ends of the culvert being studied. Thus, it is just natural to compare the numerical results with backwater calculations of the free-surface.

A backwater flow calculation assuming the culvert's outlet as hydraulic control, along with the results obtained from the CFD simulation, is presented in Figure 13. Also in the Figure, for reference, the hydraulic grade-line (or Head) and the critical depth line are plotted (details on their meaning are given in Chow (1964)). Notice that by assuming the flow is controlled by the outfall no drowning occurs in the inlet, theoretically speaking. The difference between the depths obtained by the models at the culvert's outlet may be explained by the assumption made in hydraulics that flow curvature is negligible: pressure distributions are always assumed linear, but that's not the case near an outfall. However, by further inspecting the free-surface near the inlet it is clear that the profile calculated using the Direct step method sits very close to the crown of the pipe, that is,

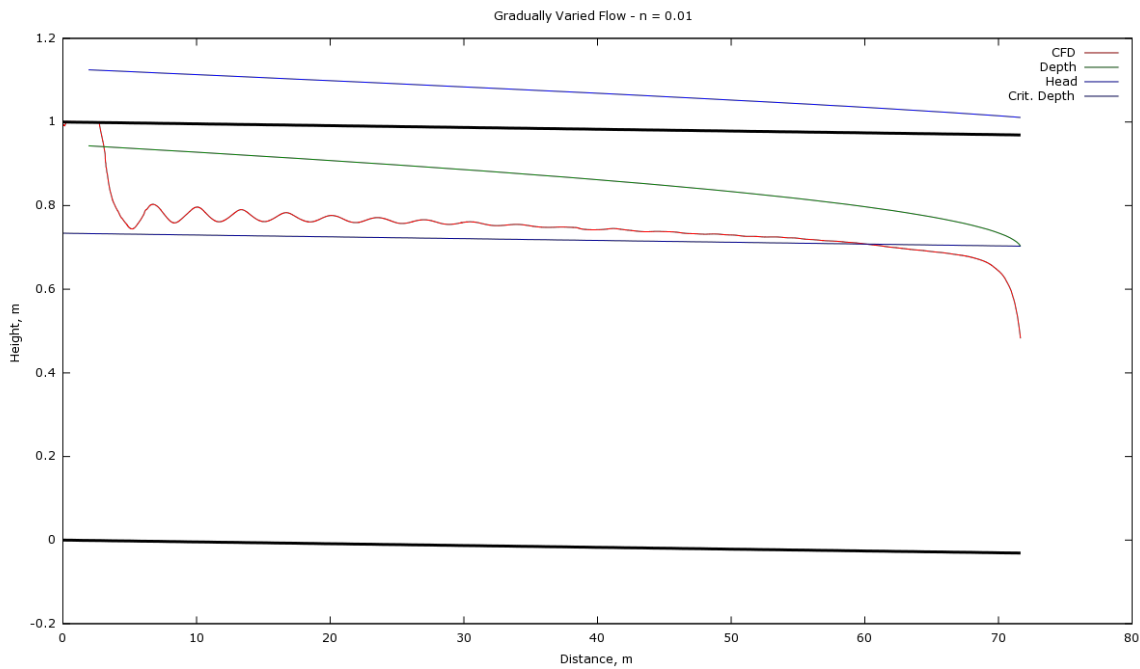


Figure 13 – Free surface inside the culvert. The solid black lines are the culvert's bottom and crest.

the pipe is around 95% filled with water near the entrance. Any disturbance, in the form of waves, may well force the free surface to attach to the upper boundary thus producing a drowned flow. From this, it is clear that the methodology of Bodhaine (1968) may offer only an approximation of the discharge in this case, and its results should be interpreted with care.

Note that, as long as the flow in the culvert exits in an outfall, geometrical modifications in the downstream section have little effect in the free surface inside the culvert. From this point onwards, all the domain upstream of the culvert's outlet is disregarded in subsequent calculations. The water height at the new inlet will be assumed equal to the critical depth, and the discharge equal to $1.5 \text{ m}^3/\text{s}$.

4.2.2 Navigability Assessment

The transversal velocity profiles at different streamwise sections are depicted in Figure 14. Note that the flow entering the channel remains mostly on the first tens of centimetres below the water surface and pours to the main channel at around 15 m downstream from the culvert. The flow encompasses an important portion of the right bank of the main channel. However, the transversal flow is only weakly present at the mid-axis streamwise plane. In general, velocities near the free surface are well above the 0.3 m/s limit, difficulting navigation.

The transversal velocities caused by the discharge from the culvert are too high, but these remain confined to a relatively thin portion of the water depth. A more precise account of the effect of such velocities on the navigation of vessels with different drafts is carried out.

Using the technique proposed in Section 4.1.4, Figure 15 is produced for each of the streamwise planes and different drafts. The drafts of vessels chosen for the analysis are meant to represent typical cases crossing the lock, that is, vessels passing with drafts of 50 cm, 1 m, 2 m, and 2.3 m. Notice that for the mid-axis streamwise section (+0.00 m offset) averaging over increasing drafts does not affect much the final profile, thus further validating the notion that transversal velocities become weaker near the mid section of the channel. On the other hand, the proposed averaging show non-negligible differences for different drafts between 15 and 35 m downstream of the culvert's outlet.

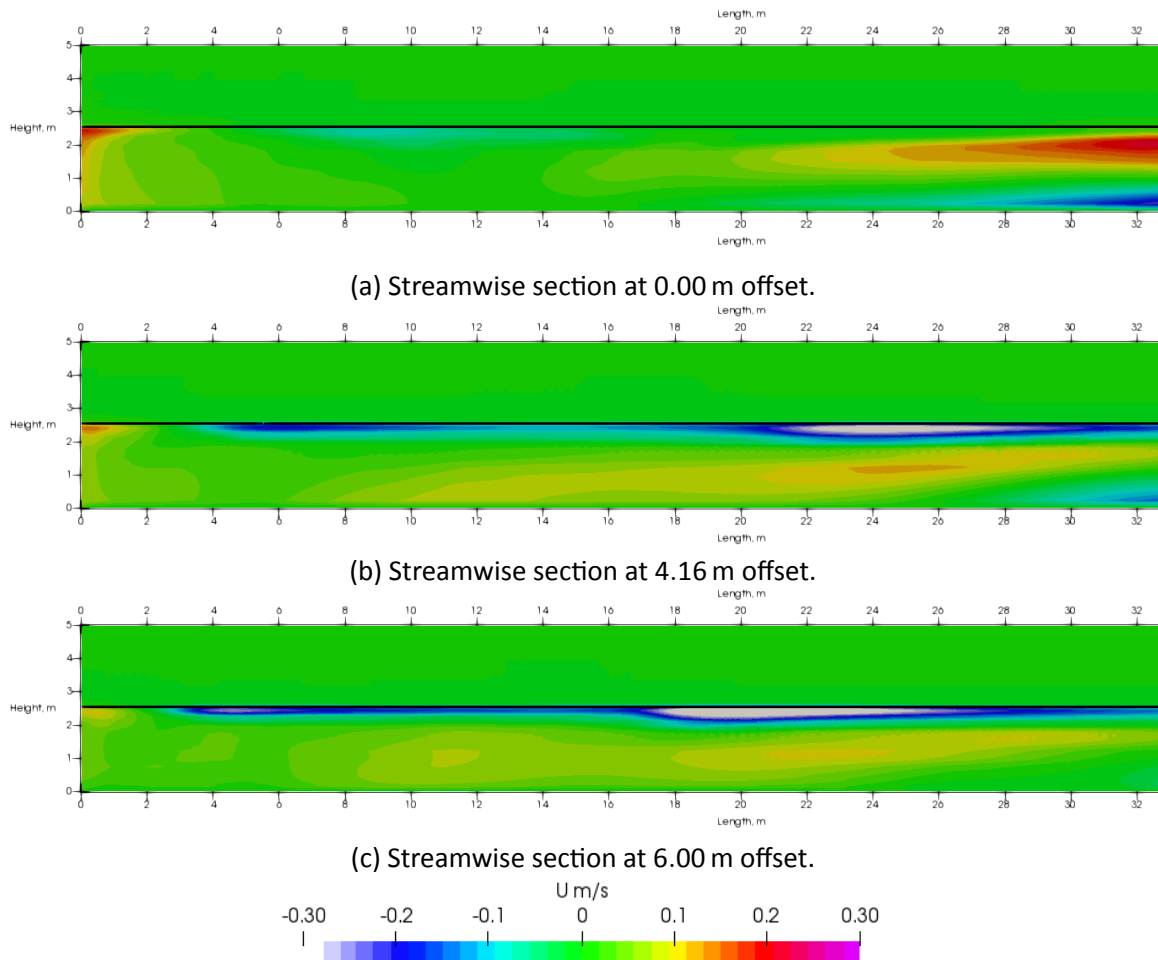


Figure 14 – Streamwise section colored by transversal (plane normal) velocities.

One may be tempted to assume that as the draft of the vessel becomes larger, navigation is less affected by the transversal flow. In reality, the disparity in the velocity profiles at different abscissas (at around 25 m) may produce a roll moment towards the port side in the vessel. Such non-uniformity in the flow is undesirable.

4.3 Numerical simulation of design alternatives

In the previous chapter it was shown that discharge from the side drainage may produce undesirable side currents when no outlet structure is present. For vessels traversing the lock while operating the side drainage these flows may impede their passage and increase the risk of collisions.

There are several counter-measures that may be adopted, either passive (i.e. traffic control, controlled discharges) or active (i.e. infrastructure), that may prove viable. On the one hand, by channelizing the side flow one may dispense of any erosion protection measure that may take place in the case where the slope where the outflow runs off is left as it is, while allowing for the addition of energy dissipation measures to control the side currents in the main channel. On the other hand, one may argue that by using traffic-signalling as a measure while regulating mass flow in the lock system may prove effective to avoid unwanted navigation risks downstream of the lock, plus by controlling the release from the drainage using valves one can mitigate the erosive forces by lowering the velocities being released.

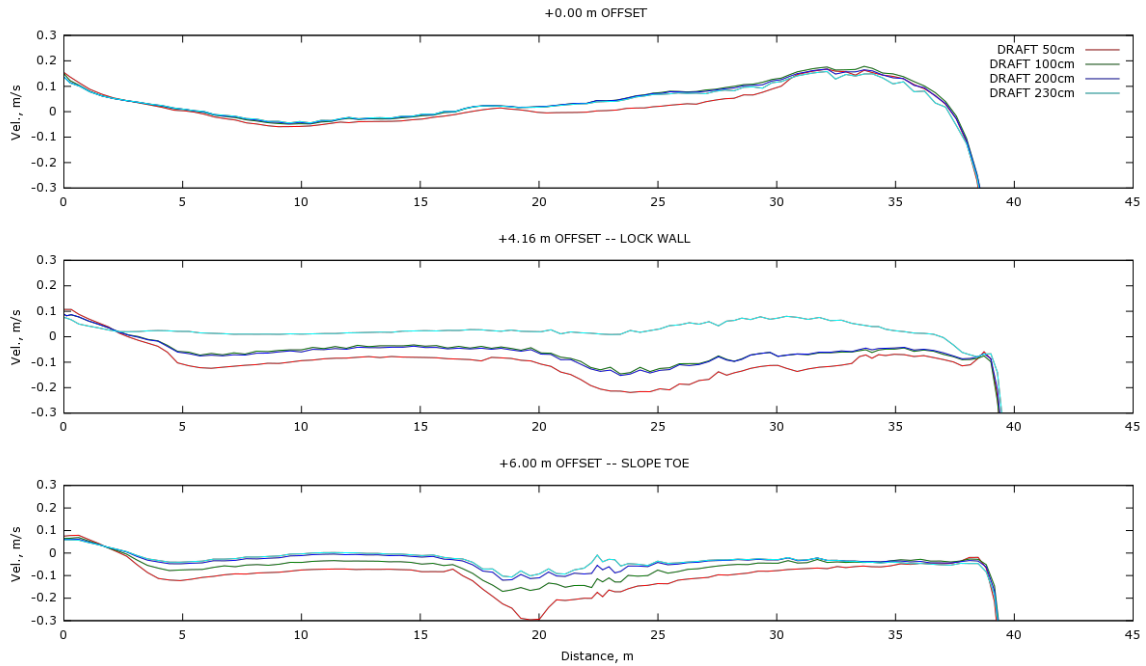


Figure 15 – Draft-averaged transversal velocities along the streamwise sections.

This section will focus on proposing active measures to mitigate the impact of an uncontrolled⁶ discharge through the side drainage. Basically, the measure considers the construction of a side channel parallel to the lock's main channel. Such side channel may deliver the flow to the main canal. The effectiveness of each proposal will be evaluated with the use of the CFD and the post-processing tools used thus far.

4.3.1 First alternative: side channel with sill

A first approach to the design of the side channel is the following: (1) one should allow the vessel to navigate undisturbed for a certain distance as to allow a portion of it to be out the lock, and (2) guarantee a certain degree of uniformity in the flow discharging onto the main channel as to avoid unwanted yaw motions onto the vessel.

The geometrical configuration of this alternative is shown in Figure 16. The channel is 24 m long and 3 m wide, fitted with a sill in order to spread the flow jet onto the whole water depth. Part of the energy incoming from the outfall is dissipated by a drowned hydraulic jump produced when the outfall flow meets the water body in the channel. The side channel's bed is one meter above the side channel, hence the side channel has a depth of 1.5 m.

The transversal velocity fields extracted from the simulation are shown in Figure 17. Note that the transversal flow tends to homogenize as one gets closer to the mid-axis of the channel. On the other hand, flow near the free surface *seems* to be recirculating to the bottom of the channel. This seemingly counter-intuitive phenomenon is better understood by looking at the flow streamlines shown in Figure 17(d). Note that the flow exiting from the channel remains lifted from the bed of the main channel, while remaining confined to the water body surrounding the right bank. The transversal velocities seen on Figure 17(a) correspond to a secondary current induced by the main flow exiting the side channel. However, the transverse velocities seem too high and rather non-uniform.

⁶Fully open knife gate.

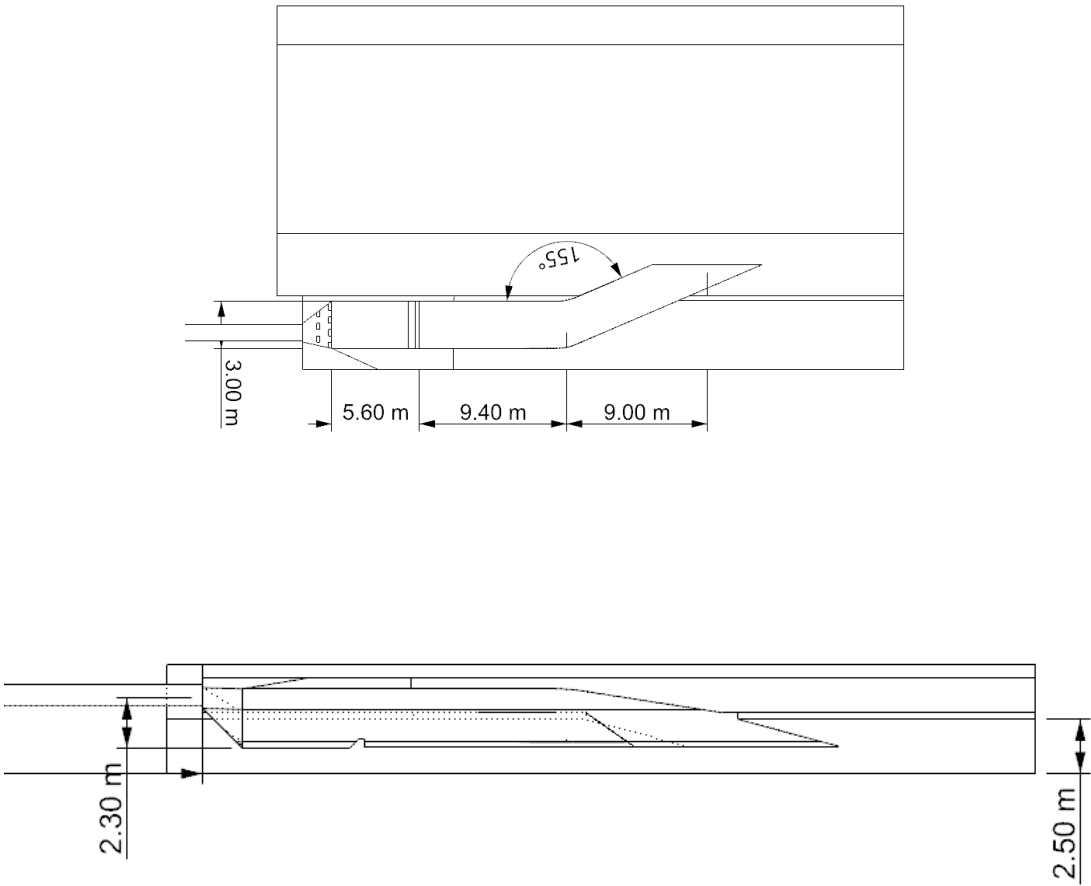


Figure 16 – Geometry and dimensions of first alternative.

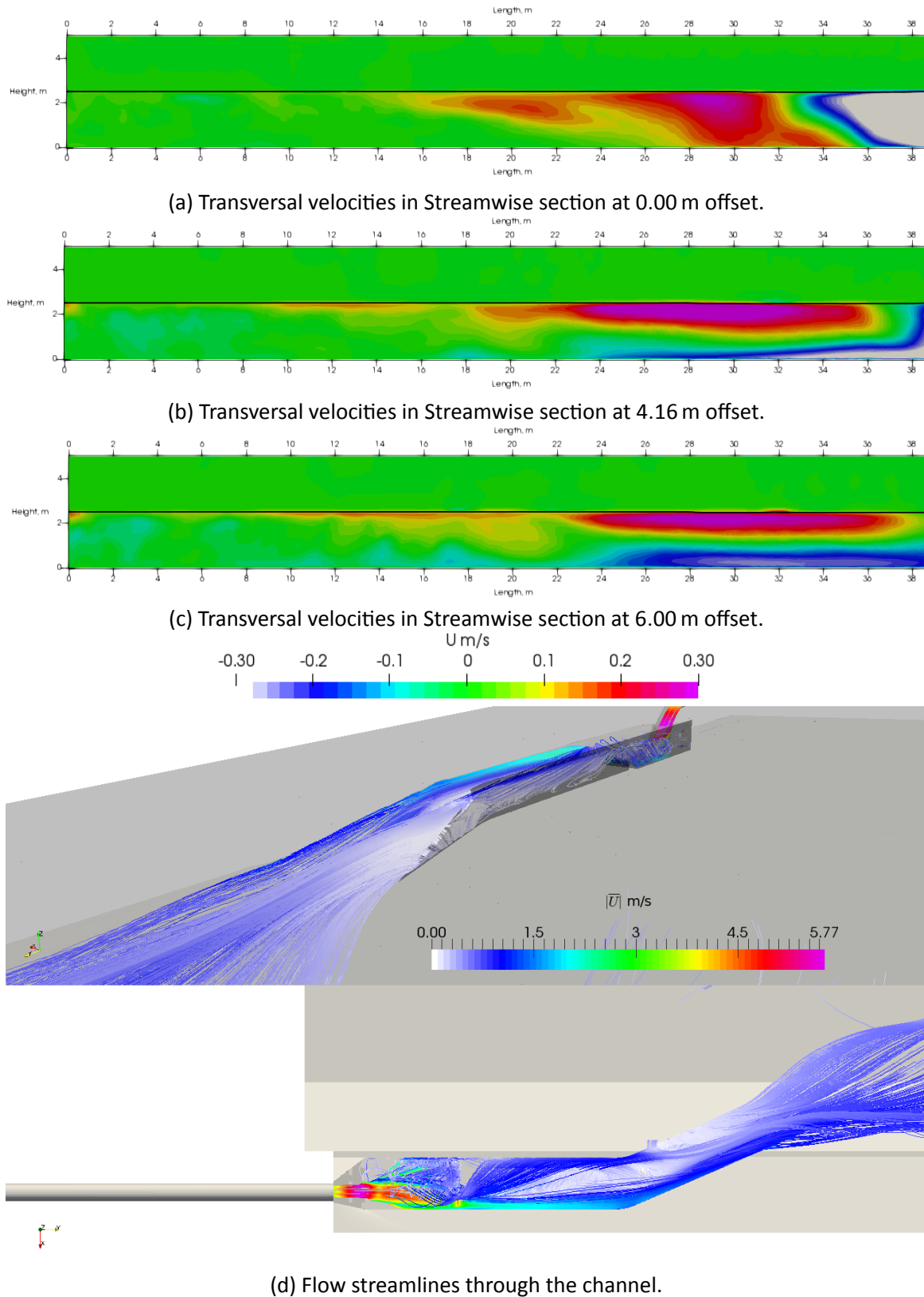


Figure 17 – Simulation results for first alternative.

A quantitative estimation of the transversal velocities along the different sections is portrayed in Figure 18. There, it is shown that: (1) the transversal velocities go beyond the 0.3 m/s threshold at the channel axis' abscissa 27.5 m, which is half the LOA of a ECMT vessel, (2) averaging over different drafts show very irregular

transversal velocities which may be undesirable, and (3) a strong recirculation region may be present and caused by the flow incoming from the channel, as it can be seen by the negative velocities from the +0.00 m offset plot.

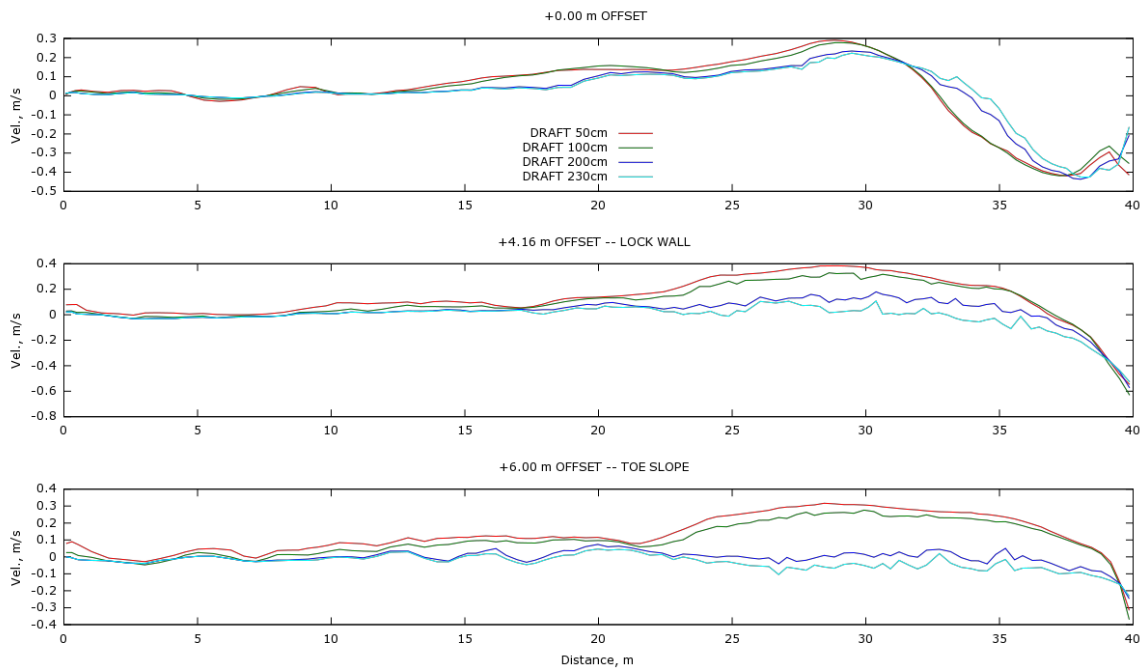


Figure 18 – Draft-averaged transversal velocities along the streamwise sections.

In this particular scenario, the total length of the downstream channel was set to 50 m. The length was chosen rather arbitrarily, in the sense that it's difficult to establish how far the outlet boundary condition should be to avoid influencing the flow in the region of study. Notice that the location of the outlet depends not only on physical or mathematical constraints, but also on computational capacity: the farther downstream the outlet is, the bigger the underlying grid. From Figure 18 it is clear that the outlet starts affecting the solution at abscissa 38 m. Although this noise does not degrade the solution nor affects the analysis just made, subsequent simulations will locate the outlet farther downstream. A channel of 75 m will be considered.

4.3.2 Second alternative: side channel with stepped slope

As listed previously, there are some features on the flow obtained in the previous alternative that are undesirable from a navigational point of view. One of them, the non-uniformity of the flow is there due to the difference in bed elevations for the side channel and the main channel. Such geometrical misalignment proves inconvenient from a hydraulic point of view. Another part of the side channel that needs attention is the right bank at the confluence: from Figure 16(d) streamlines with relatively high velocity tend to concentrate onto the right bank of the side channel, suggesting the need of erosion protection measures at the confluence towards the main channel, where the side banks may be eroded. Additionally, a smooth transition of the right bank is desirable according to what the previous scenario suggests: flow exiting the side channel tends to attach to the right bank instead of going perpendicular to the main channel.

The aforementioned features call for improvements in the previous alternative. On one end the side channel is provided with two bed elevations and making a transition using steps. The section of the channel downstream from the stepped slope has the same bed elevation as the main channel while the section upstream keeps the bed elevation proposed for the first alternative. By setting the stepped slope there is no need for a sill: energy dissipation will be induced by the drop. Additionally, a smooth transition from the right bank of the side channel to its counterpart is proposed as a way to mitigate the erosive forces caused by the flow concentrating

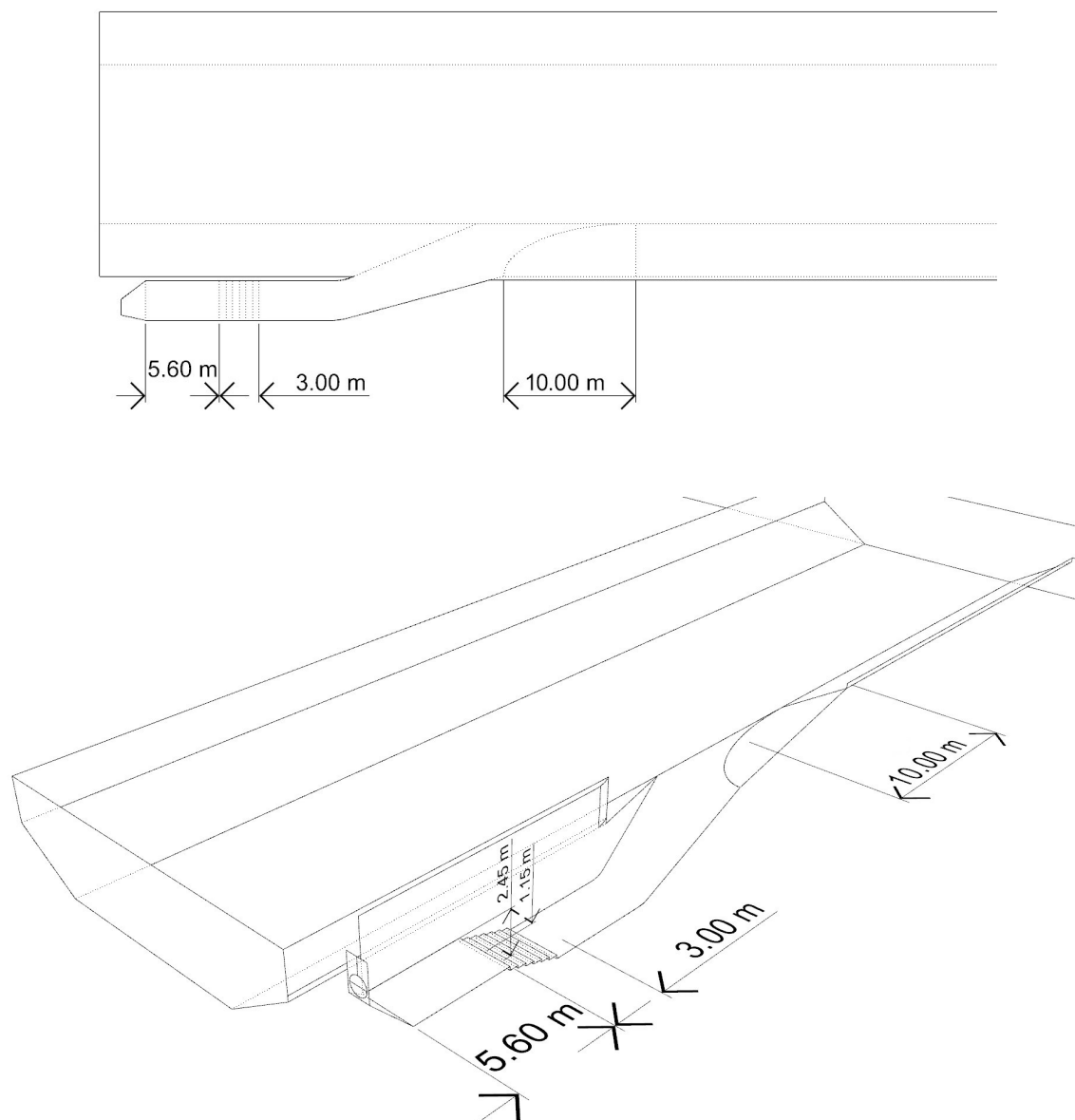


Figure 19 – Geometry and dimensions of second alternative.

onto the right bank, while smoothing the transition to the main channel's right bank. All these features are added to the previous alternative and portrayed in Figure 19.

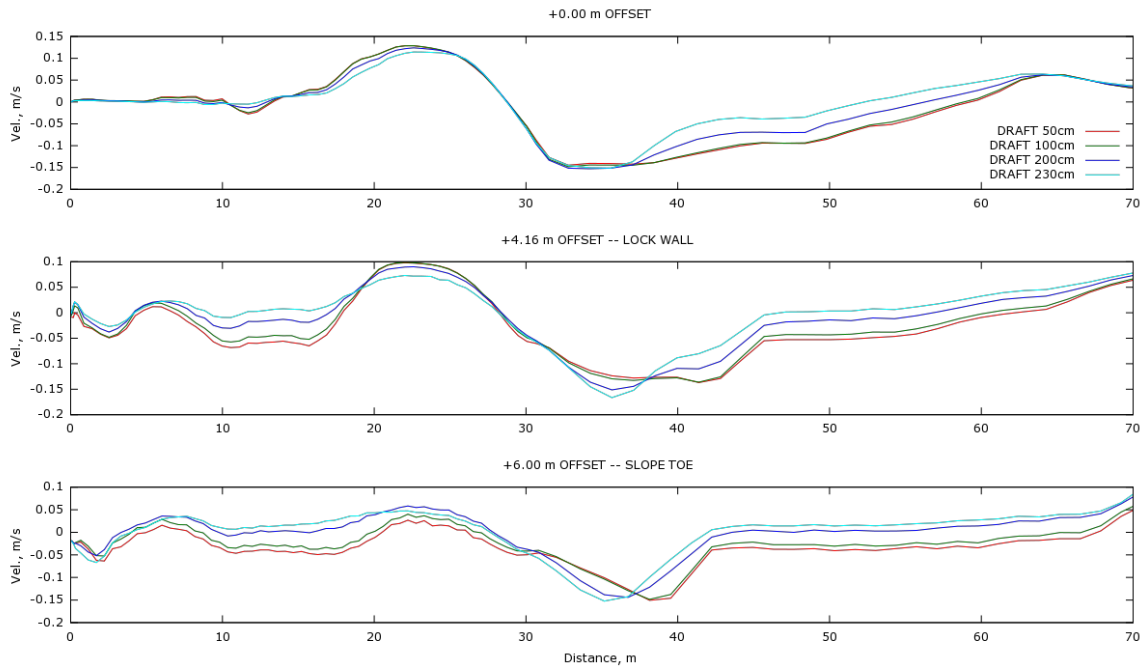


Figure 20 – Draft-averaged transversal velocities along the streamwise sections.

The present alternative shows more uniformity as well as lower transversal velocities, as it is shown in the streamwise planes of Figure 22 and in the streamline flow in Figure 21. From the three streamwise planes it is clear that a large recirculation region is present, that encompasses well beyond the mid axis of the main channel. However, velocity contours do not exceed the 0.3 m/s limit thus posing no impediments for navigation downstream. Notice that when a vessel is passing through while the side discharge is operating, this recirculation is unlikely to happen since the flow incoming from the side channel will be blocked by the submerged part of the vessel. In such an event, it is more likely that part of the flow will be directed downwards and along the starboard side of the vessel. How much of the flow goes downward or sideways will depend on the draft of each vessel.

A more accurate estimation of the impact of the velocity on the ships hull is achieved by applying the post-processing tool previously proposed (see section 4.1.4) and shown in Figure 20. As in the previous alternative, the core of the side flow appears to be around the abscissa 35 m, where the maximum transversal velocity is present at slice +6.00 m offset. Moreover, the streamwise profiles of transversal velocity are roughly insensitive to averaging depth, that is, the flow behaves more or less uniformly over the water column. As expected, the flow tends to become more uniform as one gets nearer to the mid-axis of the main channel. Finally, notice that the maximum draft averaged lateral current speed is limited to 0.15 m/s, representing only 50% of the maximum cross current prescribed by design guidelines (0.3 m/s).

Note that erosion protection measures may be needed at the bed and banks of the main channel downstream of the confluence. Several techniques exist for the dimensioning of rip-rap and rock revetments for erosion protection (Garcia, 2008). Here, we will use the shear stress method and Pilarczyk's method. The first method is based on *first principles*, that is, the calculation of the effective diameter of hypothetical well packed silica spheres covering the bed of the channel stems from the balance between the restoring and lifting forces acting on each sphere. The second methodology is an exercise in dimensional analysis, where the calculation of the effective diameter relies on the estimation of several non-dimensional parameters. Note that the calculations shown next only account for the flow in the side channel. A more complete assessment is needed in the case of a vessel passing along the channel.

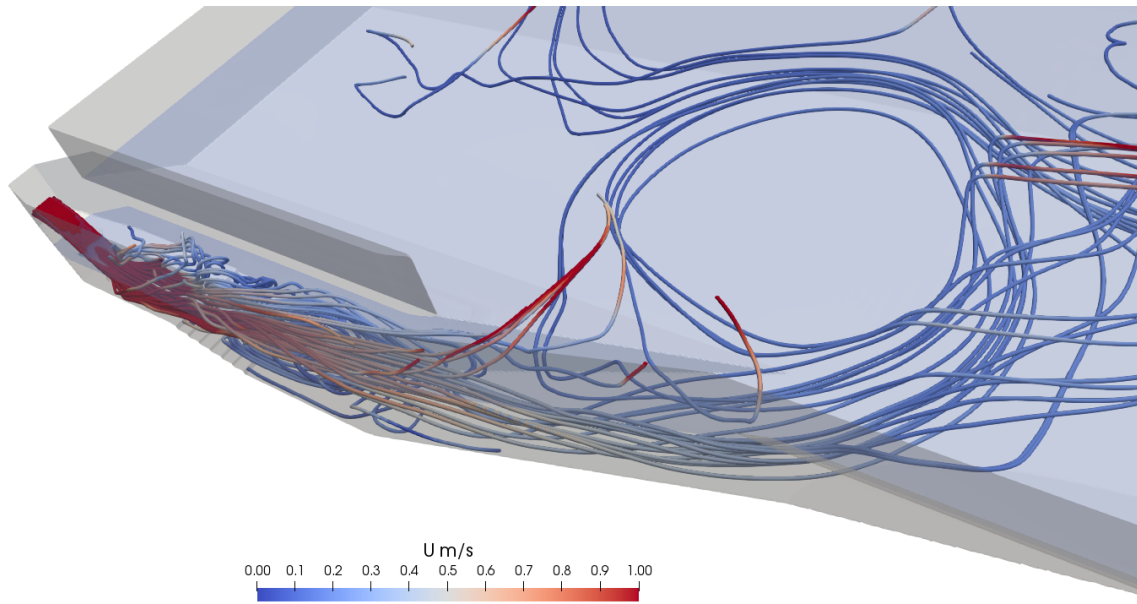


Figure 21 – Volume streamlines of the mean flow around the confluence.

The first method, the stress method (Julien, 2000), is used to produce Figure 22(d), where a top view of the main channel's bed coloured by the effective diameter is shown. There, the time-averaged bed-normal stresses are used to prepare the plot. This information may be used for the dimensioning of rip-rap (or other) measures for erosion protection. As an example, the effective rip-rap size d_m for the maximum stress may be determined using the following relationship:

$$d_m = d_{50} = \frac{\tau_0}{\tau_{*c} \gamma (G - 1) \left[\sqrt{1 - \frac{\sin^2 \theta_1}{\sin^2 \varphi}} \right]},$$

where τ_0 , τ_{*c} , γ , G , θ_1 , and φ are the shear stress, the critical Shield's shear stress ($= 0.047$) of the rocks being deposited, the specific weight of water, the specific gravity of sand ($= 2.65$), the slope of the channel's banks ($= 32^\circ$), and the angle of repose of the riprap ($= 40^\circ$), respectively. If we assume the stress on the side banks to be the maximum shown, that is $\tau_0 = 13$ Pa, then $d_m \approx 0.027$ m. Usually, for such low levels of shear stress rip-rap might not be necessary.

The velocity method, or the method of Pilarczyk, is used to produce Figure 22(d), where the top view of a horizontal plane 20 cm above the channel's bed is coloured with velocities. The method specifies the following relation of the calculation of the effective diameter d_m :

$$d_m = d_{50} = f \left(\frac{h}{k_b}, \Phi, K_t, \beta \right) \frac{1}{\sqrt{1 - \frac{\sin^2 \theta_1}{\sin^2 \varphi}}} \frac{\tau_* u_r^2}{\tau_{*c} 2g} = \frac{\Phi K_t \beta}{\sqrt{1 - \frac{\sin^2 \theta_1}{\sin^2 \varphi}}} \frac{\tau_* u_r^2}{\tau_{*c} 2g}, \quad (21)$$

where $\tau_* = 0.035$ is the reference critical Shield's stress, $\Phi (= 1.0)$ a form factor that accounts for the aspect ratio of the rip-rap's elements, the coefficients $K_t (= 1.5)$ and $\beta (= 1)$ account for turbulence and corrects for the uniform velocity assumption, and the reference velocity u_r is a characteristic velocity for which the rip-rap will be dimensioned. The critical shear stress $\tau_{*c} (= 0.035)$ is chosen according to Pilarczyk (2000), which differs from the criterion set earlier for the shear stress method. Since the characteristic velocity is chosen to be the local velocity 20 cm above the bed, β is set to unity. The definition of turbulence in this methodology is too broad and lacking any physically sound explanation, hence a value of 1.5 for K_t is guessed. The location of the plane where to extract the velocities is also chosen in a rather *arbitrary* way: usually the buffer region in the boundary layer occur at 5-10% of the channel's height. However, the Author cannot confirm whether such

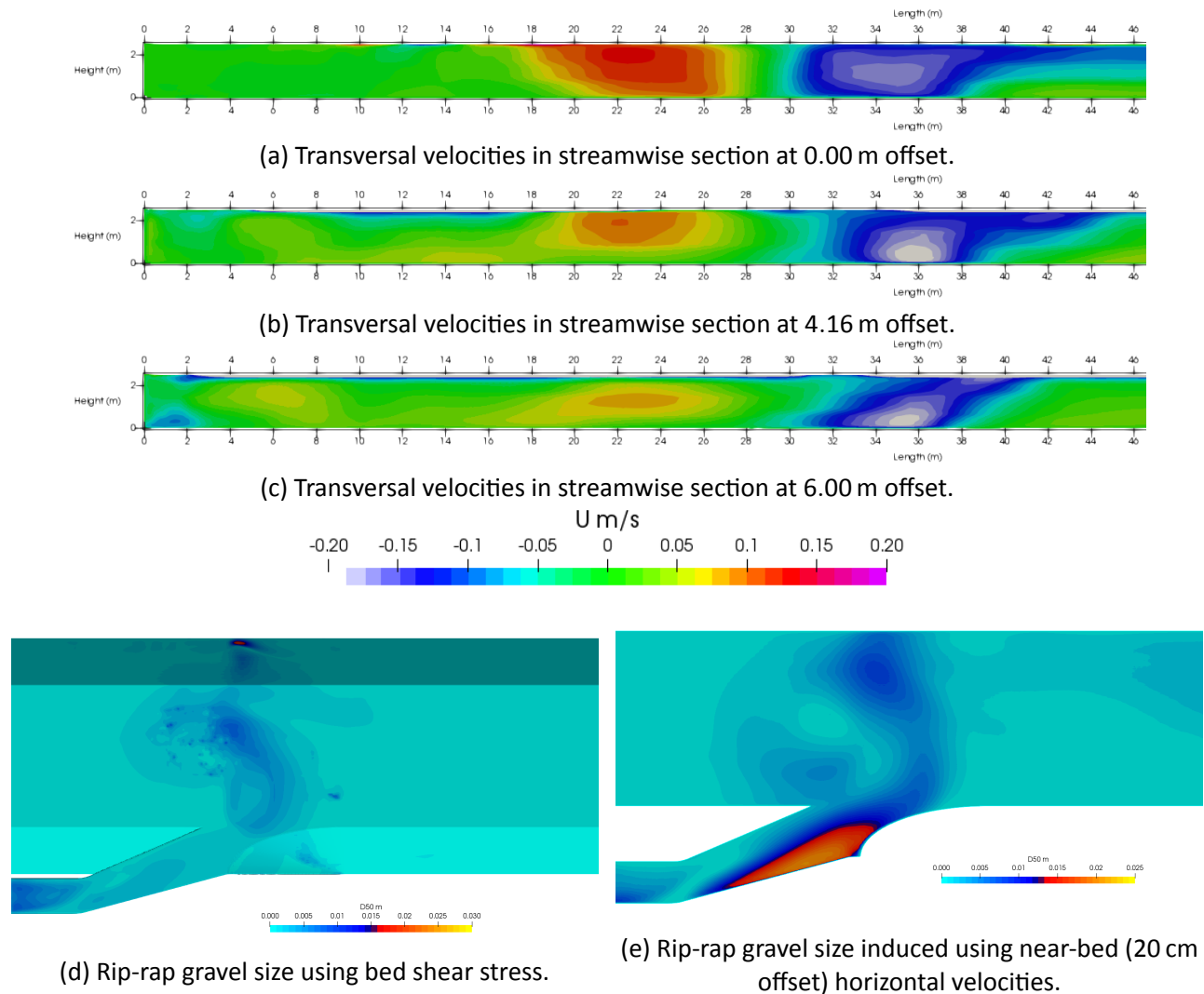


Figure 22 – Simulation results for the second alternative.

location is good for the type of calculations being made. Anyway, the maximum effective diameter calculated with this technique is $d_m=0.017$ m, which is not too different from the value obtained with the shear stress technique.

5 Discussion

The purpose of this report has been twofold: (1) determine the discharge of an existing drainage structure built on Lock No. 3 in the Dessel-Schoten canal, and (2) to propose measures for the safe operation of the side drainage while the lock is being crossed by a vessel.

The first part of this study consisted on verifying the discharge initially proposed using CFD, using one-dimensional techniques. It was found that the discharge obtained from the numerical simulations slightly differ from those obtained using a semi-empirical approach. However for the correct application of the techniques proposed by Bodhaine (1968) the required assumptions needed to be inferred from the CFD simulation. Otherwise, using the criteria proposed there, the incorrect formulae would have been used. Overall, the error in discharge determined using the corrected one-dimensional techniques and CFD was of about $0.040 \text{ m}^3/\text{s}$. The discharge thus determined is $1.5 \text{ m}^3/\text{s}$. Notice this discharge exceeds the threshold of $0.8 \text{ m}^3/\text{s}$ provided by the syphon.

The CFD calculations have shown that operating the outlet structure with a fully open valve may cause undesirable flow patterns in the culvert. Normal operations should avoid partially drowned flows in the culvert, by controlling the discharge using the valve. A further study may be needed in order to determine the optimal opening of the valve (and subsequent discharge) for which the culvert doesn't drown.

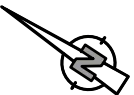
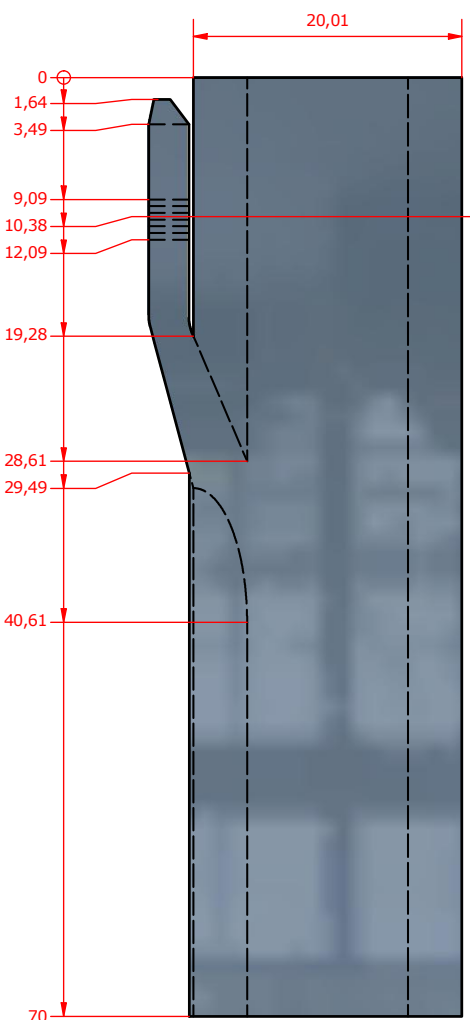
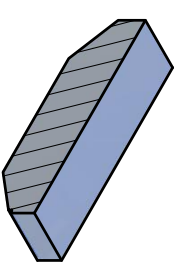
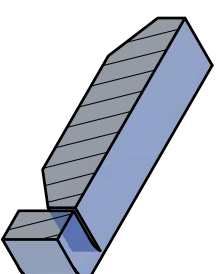
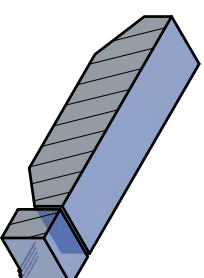
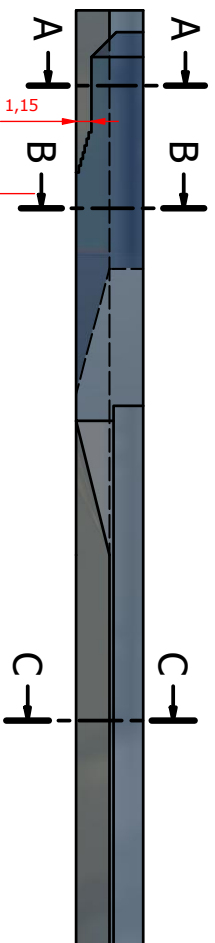
Incidentally, after studying the existing conditions in the system, by using CFD and semi-empirical tools, it was found that an outlet structure was necessary to guarantee the navigability in the main channel and stability on its right bank.

To that end, an alternative comprised by two variants has been proposed. Basically, the side discharge is channelized onto a rectangular channel of 24-by-3 meters long and wide. The first variant considers a sill within the channel and a depth of 1.5 m. Such depth proves to be less expensive both from a structural and geotechnical point of view. The second variant on the other hand, considers the same channel but with two depths: a first section 1.5 m deep and a downstream section of 2.5 m depth connecting to the main channel. The transition between the two sections is made by a stepped slope. An additional smooth transition between the channels is proposed to avoid strong shear forces on the right bank downstream of the confluence.

The present CFD study revealed that the alternative with the end sill produced a strong jet towards the right bank in both channels. Furthermore, due to the depth difference between the side channel and the main channel, the transversal flow velocities delivered onto the the main channel were strongly skewed towards the upper half of the main channel's depth. Such flow features are better avoided.

On the other hand, the alternative comprising the stepped slope shows promise. The transversal velocities in the main channel are kept well below the 0.3 m/s threshold (0.15 m/s), while keeping the flow as uniform as possible along the water column. Both objectives are achieved by deepening the downstream section of the channel and by putting a stepped transition. A technical drawing of said proposal is shown next.

The shear forces acting on the side channel and main channel banks were analysed, and shown that rip-rap cover in the right bank and confluence in the main channel might be necessary. An effective gravel size of $\approx 3 \text{ cm}$ may be considered for erosion protection, which is considered rather small. Note that the rather low shear forces produced by the flow are only erosive to channel beds composed of loose silts. In the case that the bed is composed of non-cohesive material with higher grading (sand) or cohesive soil, then erosion protection measures might not be necessary.

[illegible]

References

- Bodhaine, G.** (1968). Measurement of peak discharge at culverts by indirect methods: US Geological Survey Techniques of Water-Resources Investigations, book 3, chap. *Chapter A3*
- Chow, V. T.** (1964). Handbook of Applied Hydrology, Mc-Graw-Hill Publishing
- Garcia, M.** (2008). Sedimentation Engineering: Processes, Measurements, Modeling, and Practice. ASCE Book Series
- Julien, Y. P.** (2000). River Mechanics. Cambridge University Press
- Koedijk, O. C.; Sluijs, A. van der; Steijn, M.** (2017). Richtlijnen vaarwegen 2017: Kader verkeerskundig vaarweg-ontwerp Rijkswaterstaat. *ISBN 978-90-9030674-2*
- Pilarczyk, K.** (2000). Dikes and Revetments: Design, Maintenance and Safety Assessment. CRC Press
- Rusche, H.** (2003). Computational fluid dynamics of dispersed two-phase flows at high phase fractions. Imperial College London (University of London)
- Wilcox, D. C. et al.** (1998). Turbulence modeling for CFD. Vol. 2. DCW industries La Canada, CA

A1 On a different alternative

Yet another alternative was considered in the course of this work: a short channel with energy dissipation bars. This proposal was conceived on the basis of having a more compact structure downstream of the lock, one which required less excavation and slope stabilization. More precisely the structure is comprised of a 7.2 m long channel, equipped with vertical piers positioned in an staggered arrangement. The first three pier rows transition from a height 30 cm to a maximum of 1 m, 15 cm below the water surface. A top and side view of the structure is shown in Figure 23.

The results from the numerical simulations conducted for this scenario are shown in Figure 24. These results are illustrative of the inadequacy of the proposed alternative in lowering the transversal velocities coming to the main channel. Velocities up to 1 m/s are noticed near the free surface, roughly corresponding to the clearance between the top of the dissipation columns and the free surface.

Upon further inspection, the transversal velocities also lack uniformity along the water column. This non-uniformity as we saw previously is partly due to the depth difference between the channels. Also, as a contributor to such non-uniformity are the piers in the channel. These piers prove effective in dissipating the incoming flow along their height but also concentrate the flow towards the clearance between the roof of the highest piers and the free surface. Said clearance is of around 15 cm. The flow also presents a strong horizontal recirculation that may hamper navigation specially on low-draft and recreational vessels. An improved design may provide with taller piers, piercing the free-surface in order to also break the flow at the free surface.

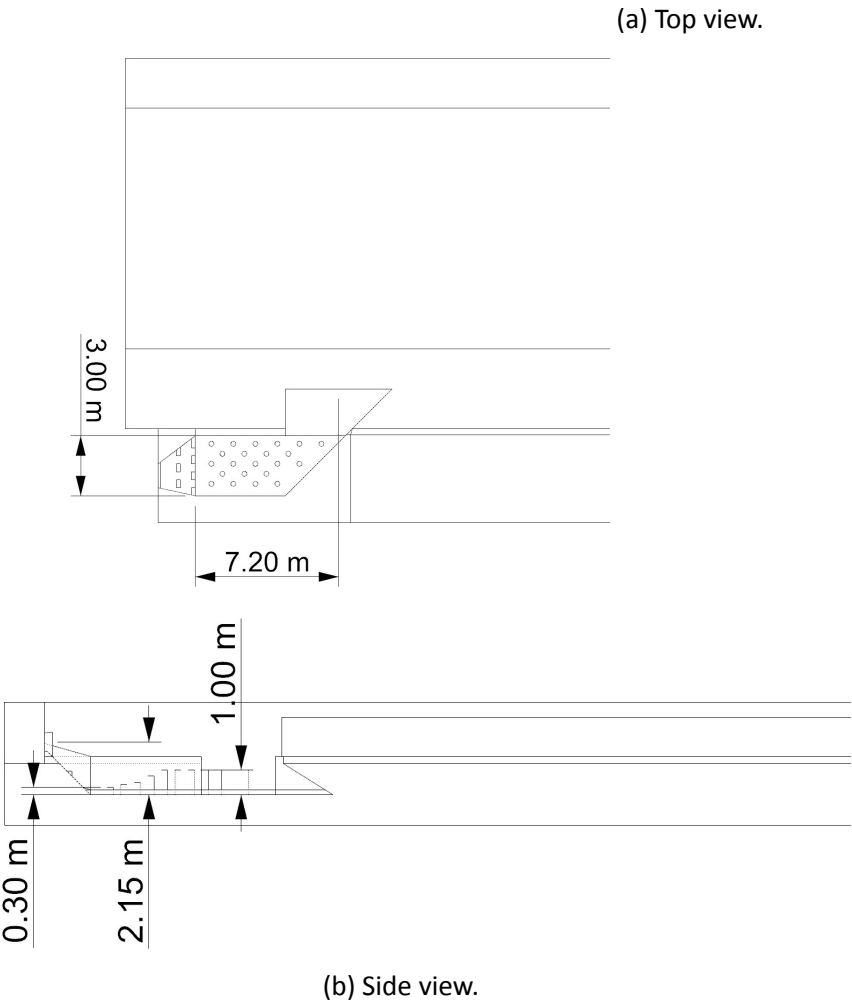
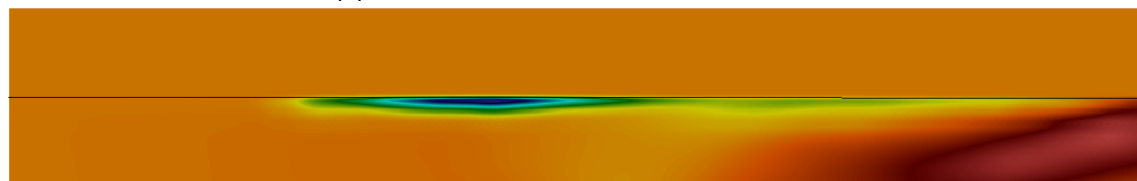


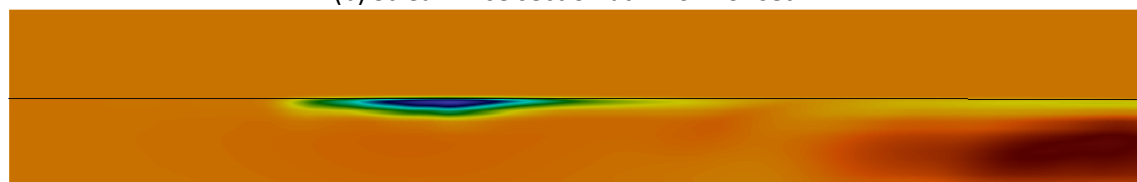
Figure 23 – Geometry and dimensions of third alternative.



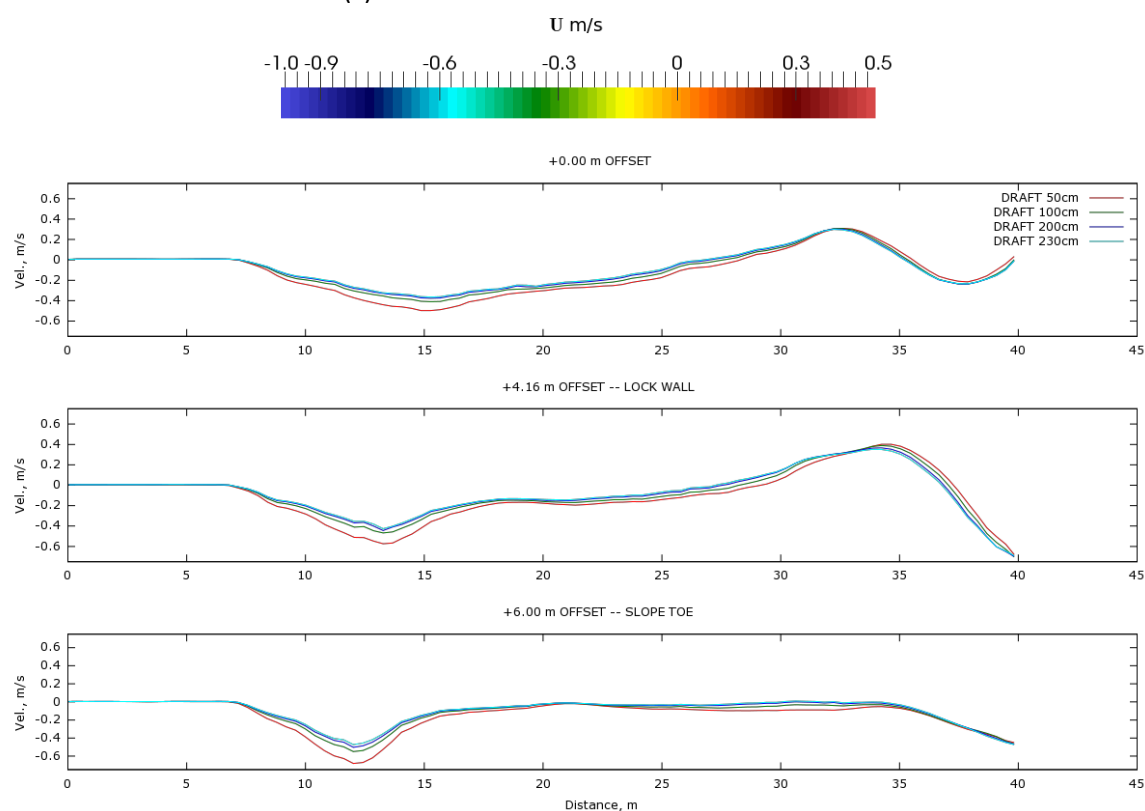
(a) Streamwise section at 0.00 m offset.



(b) Streamwise section at 4.16 m offset.



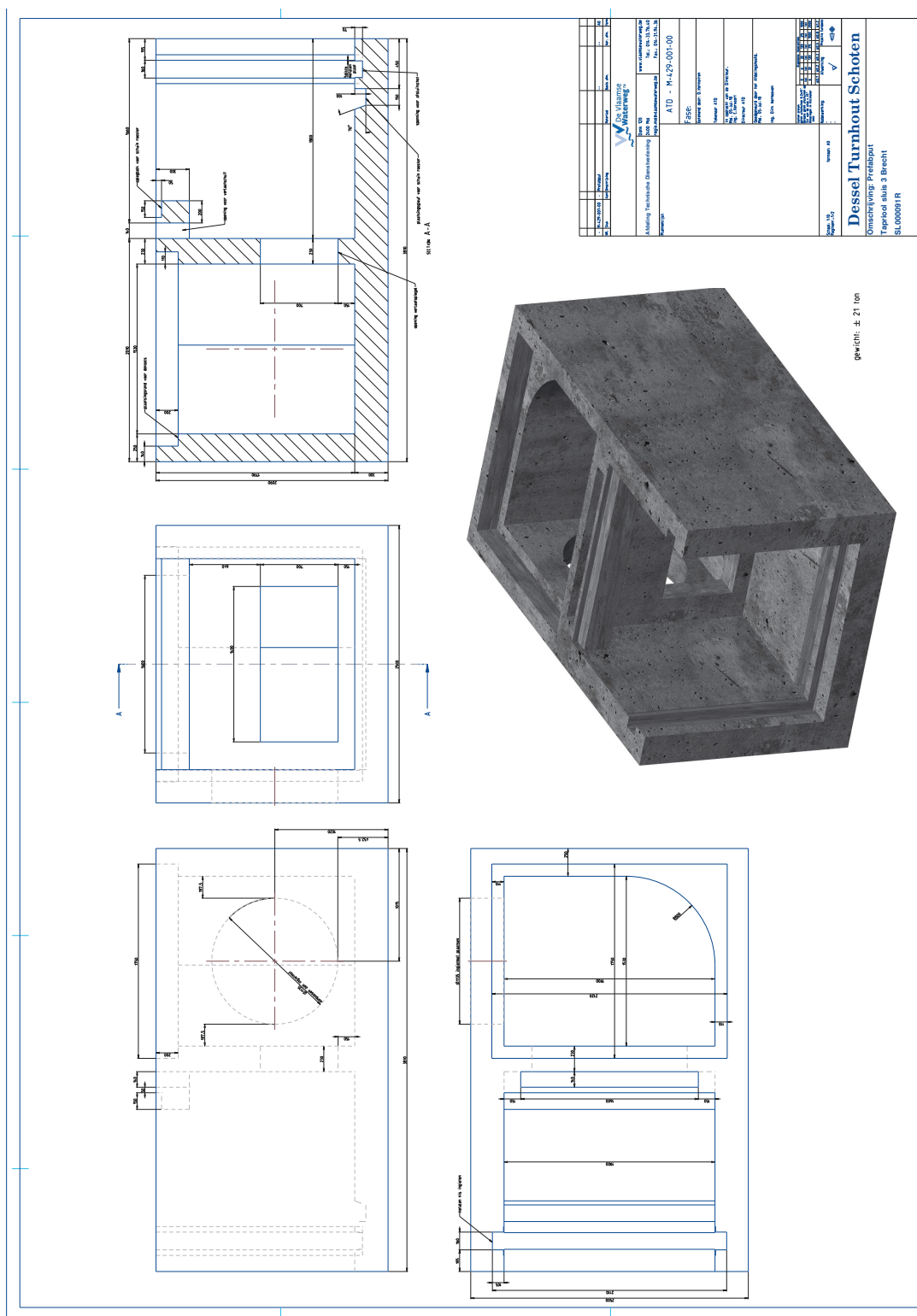
(c) Streamwise section at 6.00 m offset.

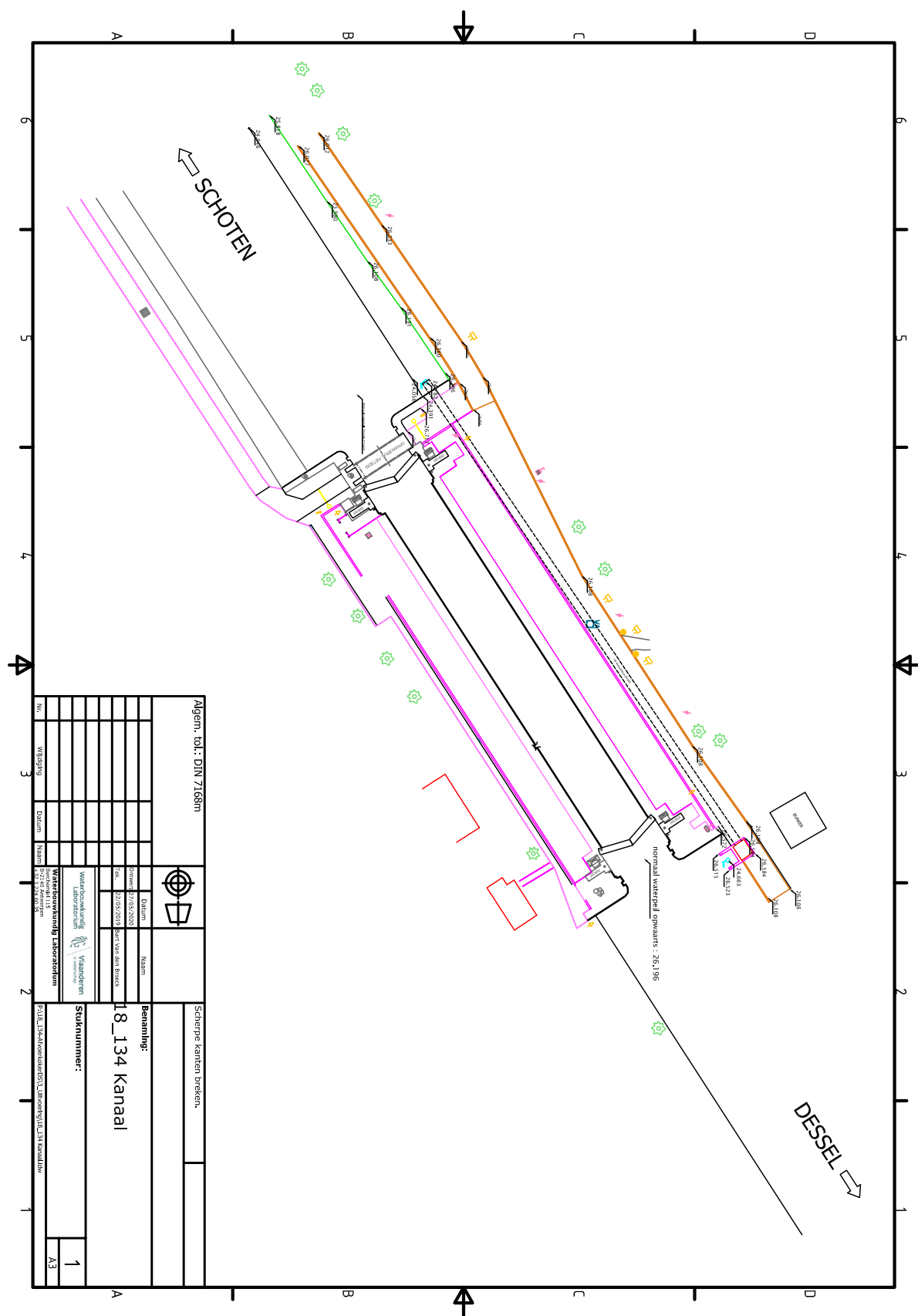


(d) Draft-average streamwise profiles of transversal velocities in the main channel.

Figure 24 – Simulation results for third alternative.

A2 Technical Drawings





DEPARTMENT **MOBILITY & PUBLIC WORKS**
Flanders hydraulics Research

Berchemlei 115, 2140 Antwerp

T +32 (0)3 224 60 35

F +32 (0)3 224 60 36

waterbouwkundiglabo@vlaanderen.be

www.flandershydraulicsresearch.be