Delft University of Technology

To Determine the Opening Width of a Navigable Weir in the Meuse by Means of Flow and Nautical Simulations During a River Flood.

Kortlever, W. C.D.; Koedijk, O.C.; Maijvis, S.D.; Zubova, A.; O'Mahoney, T.S.D.; Hove, D. ten

DOI
10.3929/ethz-b-000675980

Publication date
2024

Document Version
Final published version

Published in
Proceedings of the 10th International Symposium on Hydraulic Structures (ISHS 2024)

Citation (APA)
Kortlever, W. C. D., Koedijk, O. C., Maijvis, S. D., Zubova, A., O'Mahoney, T. S. D., & Hove, D. T. (2024). To Determine the Opening Width of a Navigable Weir in the Meuse by Means of Flow and Nautical Simulations During a River Flood. In R. Boes, I. Albayrak, S. Felder, B. Crookston, & V. Heller (Eds.), *Proceedings of the 10th International Symposium on Hydraulic Structures (ISHS 2024)* (pp. 355-364). ETH Zürich. https://doi.org/10.3929/ethz-b-000675980

Important note
To cite this publication, please use the final published version (if applicable).
Please check the document version above.

Copyright
Other than for strictly personal use, it is not permitted to download, forward or distribute the text or part of it, without the consent of the author(s) and/or copyright holder(s), unless the work is under an open content license such as Creative Commons.

Takedown policy
Please contact us and provide details if you believe this document breaches copyrights.
We will remove access to the work immediately and investigate your claim.

This work is downloaded from Delft University of Technology.
For technical reasons the number of authors shown on this cover page is limited to a maximum of 10.

ETH zürich

ETH Library

To Determine the Opening Width of a Navigable Weir in the Meuse by Means of Flow and Nautical Simulations During a River Flood

Conference Paper

Author(s):
Kortlever, Wim C.D.; Koedijk, Otto; Maijvis, Sam D.; Zubova, Anastasia; O'Mahoney, Tom S.D.; ten Hove, Dick

Publication date:
2024

Permanent link:
https://doi.org/10.3929/ethz-b-000675980

Rights / license:
Creative Commons Attribution 4.0 International

This page was generated automatically upon download from the ETH Zurich Research Collection.
For more information, please consult the Terms of use.

10th International Symposium on Hydraulic Structures Zurich, Switzerland 17-19 June 2024
ISSN 0374-0056 DOI: 10.3929/ethz-b-000675980

To Determine the Opening Width of a Navigable Weir in the Meuse by Means of Flow and Nautical Simulations During a River Flood

W.C.D. Kortlever[1], O. Koedijk[1,2], S.D. Maijvis[3], A. Zubova[3], T.S.D. O'Mahoney[3] & D. ten Hove[4]
[1]*Ministry of Infrastructure and Water Management, Rijkswaterstaat, Utrecht, The Netherlands*
[2]*Delft University of Technology, Delft, The Netherlands*
[3]*Deltares, Delft, The Netherlands*
[4]*MARIN, Wageningen, The Netherlands*
E-mail: wim.kortlever@rws.nl

Abstract: *The preliminary designs of the new weirs in the river Meuse have at least three openings because of the required redundancy during large-scale maintenance or calamities. However, considering two openings of 40 m and one of 20 m, the question arises whether these 40 m openings are wide enough to be navigable during river floods. To answer this question, first, the characteristics of the Dutch navigable weirs in the Lower Rhine and Meuse have been analyzed. Then, CFD simulations of the flow through the weir openings during a river flood have been carried out, to determine flow velocities and the eddies generated at the weir abutments and behind the pillars. The passage of the vessel through the weir opening may be greatly hindered by flow patterns with changing transversal velocities along the sailing line. The flow simulations showed that the size of the eddies near the bank and the amplitudes of the transversal velocity depend on the hydraulic design of the weir and the roughness of the bed behind the weir sill. When a realistic bed roughness is applied the influence of the horizontal eddies reduces. In the last step, for a preliminary weir design with an assumed bed roughness, the calculated flow pattern has been put into the real-time nautical simulator and simulations were carried out with a design vessel. It showed that it depends both on the hydraulic design of the weir and the wind conditions whether a new weir with a 40 m wide opening will be navigable.*

Keywords: *Weir, CFD, eddy, bed roughness, navigation, real-time.*

1. Introduction

Plans are drawn up for the renovation and replacement of the seven weirs in the Dutch part of the river Meuse. Now, five of these weirs are a combination of a Stoney-weir for the fine control, 34 m wide, and a Poirée-weir for the rough setting and large river discharges, 60 to 68 m wide (see Figure 1, left). The other two Meuse weirs have lifting gates with control valves. The total width of the river is about 100 m. Currently, four of the Poirée-weirs are navigable during river floods, when the weirs are fully opened. As the old Poirée-weirs are relatively difficult to operate and do not meet all the requirements with respect to healthy and safe working conditions, it is expected that these weirs will soon have to be replaced by modern gate types, e.g. lifting gates or segment gates. Preliminary designs presume a new weir with at least three openings between 20 and 40 m wide, because of the required redundancy during large-scale maintenance or calamities. However, considering two openings of 40 m and one of 20 m, the question arises whether these 40 m openings are wide enough to be navigable during river floods. To answer this question, first, the characteristics of the Dutch navigable weirs in the Meuse and the Lower Rhine have been analyzed.

In the twenties of the last century the Dutch part of the river Meuse was canalized in order to enable an efficient waterborne transport of black coal to the west of the Netherlands (Schot et al. 2016). This was realized by building seven weirs with water heads between 3 and 5 m (de Vries et al. 1935). The five weirs in the main navigational route (Meuse Route) were given a navigable opening with a width of 60 m, as this was the established width of the continuous navigation channel at that time. The design vessel was a Rhine vessel with a loading capacity of 2000 t (length 100 m, beam 12 m, draft 2.8 m), but also a tow of two Rhine vessels and a tugboat was considered (van Konijnenburg et al. 1912).

In the course of time the navigational channel of the Meuse has been deepened and widened, for reasons of water safety during river floods and navigation. Simultaneously, the propulsion power and the maneuverability of the vessel fleet has significantly increased. Currently, both the navigational channel and the ship locks in the Meuse Route are enlarged to accommodate push-tow units of CEMT-Class Vb, with a loading capacity of 5000 t (length 190 m, beam 11.4 m, draft 3.5 m). Likewise, these push-tow units will use the navigable opening of 60 m when the weir is fully open.

Depending on the circumstances, when two vessels encounter at the open Poirée-weir, the vessel going upstream gives right of way to the vessel going downstream. Due to changes of the river dimensions and hydraulic conditions, the flow velocities in the navigable weir opening have increased from roughly 1.5 m/s, shortly after the canalization of the Meuse, to 2.5 m/s at present.

Close to the Dutch border the Rhine splits up into three branches. In the sixties of the last century one of these branches, the Lower Rhine, has been canalized by the construction of three sets of weirs and locks. Thus, the water distribution between the river branches can be controlled and the water depths for inland navigation have been optimized. Each weir has two visor gates, mounted on three piers, one of which is in de middle of the river. See Figure 1 (right). At high river discharges the weirs will be lifted, so that the Lower Rhine is again a free flowing river (de Gaay and Blokland 1970). Then, one opening is used by vessels going upstream and the other by vessels going downstream. The flow channel between the piers has a width of 48 m. Due to the visor type of gate, the required head room is only ensured in the middle of the passage, across a width of 38 m. Also on the Lower Rhine, push-tow units of CEMT-Class Vb are allowed. The flow velocities in the weir opening, when the weir has just been entirely opened, are approximately 1.5 m/s.

Figure 1. Left: Stoney-weir (2 x 17 m) and navigable Poirée-weir (63 m). Right: visor weir (2 x 48 m).

A first estimate for the required opening width for a safe vessel passage has been obtained by applying the design rules according to the Dutch Waterway Guidelines. This has resulted in an opening width of circa 38 m. Since at the visor weirs the net width of the passage is also 38 m, it has been decided to design a new Meuse weir with two navigable openings of 38 m and one control opening of 24 m, as a starting point for further studies.

The passage of the vessel through the weir opening may be greatly hindered by flow patterns with changing transversal velocities along the sailing line. Therefore, the flow velocities and the eddies generated at the weir abutments and behind the pillars have been determined with three-dimensional Computational Fluid Dynamics (CFD) simulations. The flow field was calculated using the commercially available CFD software package Simcenter STAR-CCM+ using the Volume of Fluid (VoF) method for free surface tracking. This model has been applied previously by de Loor (2018). The flow simulations have shown that the size of the eddies near the bank and the amplitudes of the transversal velocity depend on the hydraulic design of the weir and the roughness of the bed behind the weir sill.

Finally, for a preliminary weir design with an assumed bed roughness, the calculated flow pattern from the CFD model has been put into the real-time nautical simulator and simulations were carried out with a CEMT-Vb design vessel. It shows that it depends both on the hydraulic design of the weir and the wind conditions whether or not a new weir with a circa 40 m wide opening will be navigable.

Firstly, in the next section, the preliminary design of the weir is described. The subsequent sections address the flow simulations with CFD and the real-time simulations. The studies presented in this paper have been commissioned by Rijkswaterstaat. The CFD simulations have been reported in (Maijvis et al. 2023) and the real-time nautical simulations have been reported in (Ten Hove 2023).

2. Design of New Meuse Weir

2.1. Layout and Hydraulic Conditions

The preliminary layout is shown in Figure 2. The navigable openings have a width of 38 m, the additional control opening has a width of 24 m, and the two piers in between have a width of 7 m.

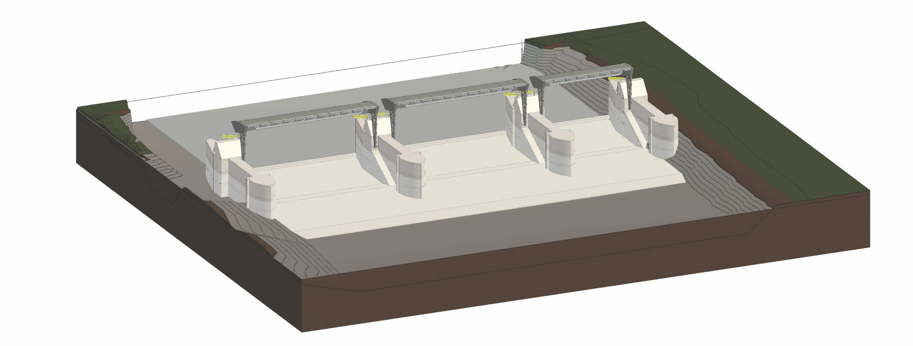

Figure 2. Preliminary layout of navigable weir, open weir with lifted segment gates (looking downstream).

One navigable opening is situated next to the left bank of the river, so that vessels going upstream can continue their path along this bank. The other navigable opening is situated in the middle, so that vessels going with the flow, which are less maneuverable, can pass in the middle and are not hindered by any bank effects. The segment gate is only an example of a common gate structure. Other solutions with vertical lifting gates are also considered. This weir design has been made fit for the weir location at Sambeek in the Meuse.

The Meuse river is fed by rainwater and melt water from the Ardennes. The average discharge of the Meuse river is about 200 m^3/s, and roughly varies between 20 m^3/s during a dry period and 3500 m^3/s at extreme events. The target water level in the upper reach of Sambeek is NAP + 11.1 m (NAP: Amsterdam Ordnance Datum). The new weir sill is situated at NAP + 4.2 m. So, the maximum head at the weir is circa 7 m. It is estimated that the weir will be fully opened at a river discharge of about 1400 m^3/s. Then, the water level at the weir is circa NAP + 10.8 m. The flood plains start at NAP + 11.90 m.

For the Meuse at Sambeek, the design vessels are an extended large motor vessel (l x b x d: 137.5 x 15.50 x 3.5 m^3) and a push-tow unit (l x b x d: 193 x 13.5 x 3.5 m^3). Because of its larger length, the push-tow unit is decisive for the required opening width of the weir.

2.2. Weir Sill Design

Figure 3 shows the outline of the weir sill. The riverbed upstream lies at NAP + 3.6 m, downstream at NAP + 3.4 m, the weir sill is at NAP + 4.2 m. Just behind the gate is a stilling basin with a depth of 1 m and a total length of 20 m.

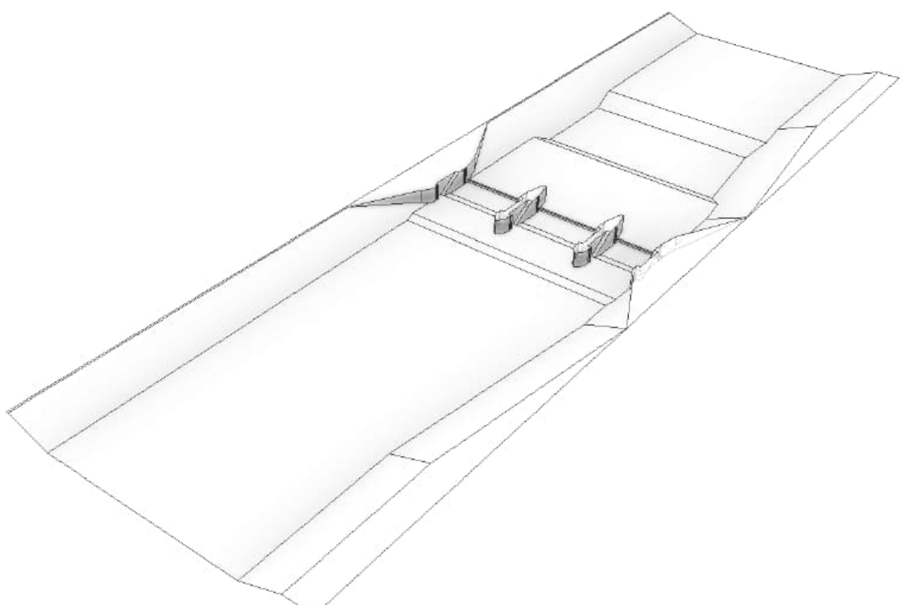

Figure 3. Isometric view of weir sill, pillars, abutments and river banks (left is upstream and right is downstream).

Since the soil below the river bed consists of layers of sand and gravel, it is foreseen that a bed protection has to be constructed directly behind the stilling sill, with a length of 60 m, and at a level of NAP + 3.8 m. It is assumed that in course of time a scour hole will develop beyond the bed protection. In the hydraulic model this scour hole has a depth of 2 m and a length of more than 50 m. For the present, the pillars of the weir are 7 m wide and 41.9 m long. The upstream face of the pillar has a sharp end, which is related to the discharge of ice, and an elliptical rounding (2 to 1). The downstream end of the pillar has a slope of 4 to 1 with fixed separation points for the flow. The wing walls at the abutments have a slope of 2 to 1 in the horizontal plane. The slopes of the river bank are 2.5 to 1.

3. Flow Simulation with CFD

3.1. Objective

The objective of the CFD simulations was to provide time-dependent flow fields, depth-averaged over the draft of the vessel, which can be used in the real-time nautical simulator. It is expected that the properties of these flow fields are determined by the hydraulic design of the weir sill, the pillars and the abutments, and by the roughness of the bed protections and the dimensions of the scour hole.

To model the effect of the river bed, the majority of CFD studies of natural streams assume either a smooth bed or use the traditional wall function approach with a roughness length. Characterizing the effects of large roughness elements remains a challenge due to the inadequacy of traditional roughness representations to characterize flow profiles in this situation (Carney et al. 2006). Despite its weak intensity, the secondary flow effects induced by the roughness elements are expected to significantly influence the flow and alter the mean velocity profile due to the generation of local vorticity parallel to the flow. In this study, to simplify the geometry of a real, irregular rock protection while still introducing the correct amount of turbulence in the model, this protection was schematized as trapezoids in a regular pattern. It is important to note that using a physically rough bed instead of wall functions requires a high-quality mesh and results in high computational costs.

3.2. Design Variations

To find the most effective design in the interest of nautical safety, flow fields were determined for several variations of the design or layout. In the initial design the wing walls have a slope of 2 to 1 and a scour hole has been added. To study the effect of streamlining the abutments an alternative slope of 4 to 1 has been simulated (Figure 4). The influence of the scour hole on the flow pattern has been estimated by carrying out one simulation without scour hole.

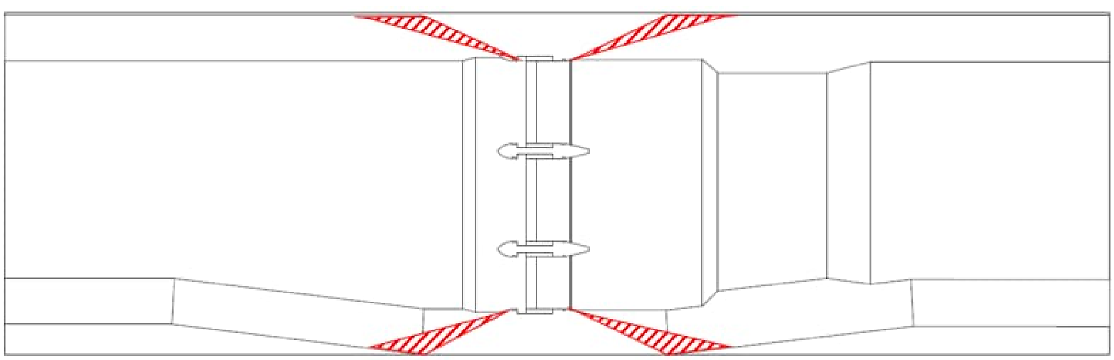

Figure 4. Streamlining the wing walls at the abutments, slope angle 4 to 1 instead of 2 to 1.

Earlier simulations without a bed protection structure had resulted in large eddies and consequently in unsafe conditions from the perspective of nautical safety. A schematized bed protection structure has been added to the model in order to increase the accuracy of the simulated flow fields. The original bed protection design is based on a broad overview of the bed protections applied at the current weirs in the Meuse and in the Lower Rhine. Two different schematizations of the bed protection have been considered: trapezoids in either a ‘staggered’ or a ‘checkerboard’ pattern (Figure 5). The base width of the trapezoids is 0.35 m, and the height of each element is 0.15 m. In the checkerboard pattern there is an additional height difference of 0.15 m between patches of 4 x 4 trapezoidal elements, which is expected to introduce an increased flow roughness. The bed protection is located behind the weir and extends 10 m onto the river banks.

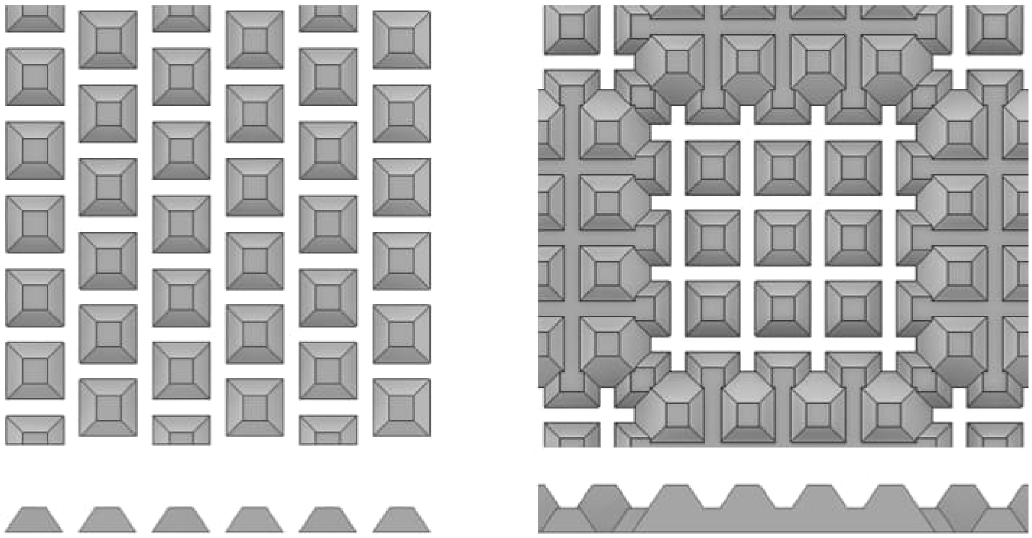

Figure 5. Top and side view of the bed protection patterns used in the CFD model: Staggered (left), Checkerboard (right).

3.3. Model Set-up

The CFD models of the new weir design and its variations were set up using the commercial software package Simcenter STAR-CCM+ 2021.2 Build 16.04.007. Air and water were modelled as immiscible phases by employing the Volume of Fluid (VOF) method. The mesh was generated using STAR-CCM's built-in trimmer mesh algorithm, resulting in predominantly square control volumes with prismatic shapes near boundaries, and where necessary, a so-called prism layer to capture the velocity gradients near the solid boundaries. The imposed prism layer had a thickness of 0.114 m and consisted of 7 layers with a layer stretching factor of 1.4. Therefore, the cell resolution adjacent to the solid walls of the riverbed and the weir structure was sufficiently fine to capture boundary layer effects arising from adverse pressure gradients. The overall base size of the cubic cells was equal to 0.7 m in all directions. From there, the mesh gradually became finer towards the regions of interest. A refinement region with a resolution of 0.6 m in all directions was applied near the weir structure. Near the bed protection and the scour hole, the resolution was refined to 0.2 m in all directions. Lastly, to resolve the head difference over the weir, a refinement region was added near the free surface that consisted of cells with a vertical resolution of 0.1 m and horizontal resolution of 0.7 m. On average, the total number of cells in each of the performed simulations was approximately 50 million. A limited mesh sensitivity analysis was performed using the simulation with the staggered bed protection schematization by applying different cell sizes near the weir and in the other regions of interest (free surface, bed protection and scour hole) to ensure that the cell size was sufficient to capture the relevant flow phenomena. Since halving the cell size around the sill of the weir did not change the contraction over the sill, it has been concluded that the gradients in this region were sufficiently resolved using the less fine mesh.

The turbulent fluid flow was modelled using an Unsteady Reynolds-Averaged Navier-Stokes (URANS) approach. This URANS approach can resolve large-scale variations in time and space and is therefore sufficient given the typical time and length scales needed for the nautical assessment based on the characteristic flow fields. The time step used in the URANS simulation was equal to 0.1 s. STAR-CCM+'s implicit solver does not require a Courant number smaller than 1 for numerical stability. In this case, the Courant number at the weir and the bed protection was around 1, never larger than 2, and in the rest of the domain smaller than 1. The simulations took approximately one month of computing in real-time, which corresponds to two hours of modelled flow.

The realizable k-ε model was used to model the turbulence. When using a turbulence model, the physics of the turbulent boundary layers, such as those near the riverbed and structures, must be modelled explicitly. Curvature correction was applied to the turbulence model in order to resolve turbulent recirculating flows. In this set-up, the thin viscosity-affected layer near solid walls was defined by an empirically derived wall function that satisfies the physics in the near wall region. The flow at the inlet boundary was a fully developed, turbulent and incompressible freshwater discharge of 1400 m^3/s. The height of the free surface was enforced at NAP + 11 m and NAP + 10.75 m at the inlet boundary and outlet boundary, respectively. In the region downstream of the inlet, but upstream of the outlet, the water level was calculated by the model.

3.4. Results

The 3D and high resolution nature of the simulations allowed flow features such as vortices and recirculation zones to be resolved, which could be of importance to nautical safety. In this section, the results of the CFD simulations are presented for all design variations considered. The differences between these variations are summarized in Table 1.

Table 1. Summary of the four simulated variations.

Simulation	Bed protection schematization	Wing wall slope	Scour hole present
Staggered	Staggered	2 to 1	Yes
Checkerboard	Checkerboard	2 to 1	Yes
Wing walls 4:1	Staggered	4 to 1	Yes
No scour hole	Staggered	2 to 1	No

3.4.1. Head difference

The simulated free surface height along the center line of the weir is shown in Figure 6 for all design variations. For both schematizations, bed roughness leads to a smaller decrease of the free surface level above the area with bed protection. In the model without a scour hole, the water level at the location of the scour hole is lower. This is caused by the absence of the flow deceleration, associated with higher water levels following Bernoulli's principle, which would occur in this region when a scour hole is present. It is noteworthy that the bed protection of trapezoidal elements does not significantly increase the water level upstream of the structure. So, the higher roughness that is introduced by the elements does not increase the head losses across the whole structure. Adjusting the angle of the wing walls leads to lower head losses. Similarly, removing the scour hole also reduces the head loss over the structure as the deceleration zone above the scour hole (and subsequent acceleration) is eliminated.

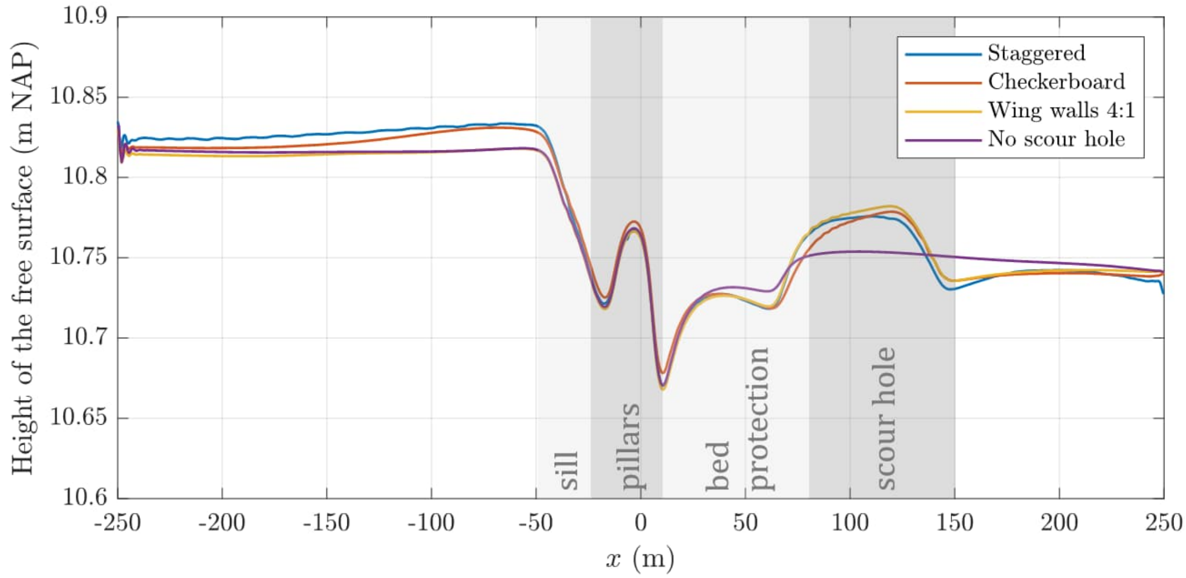

Figure 6. The height of the free surface along the center line of the weir for each of the design variations.

3.4.2. Flow behavior

The CFD simulations show that vortices develop near the left bank just downstream of the weir in the set-up with the staggered bed protection (Figure 7). Figure 8 shows the maximum transverse velocities which occur along the draft of the design vessel over time at locations A, B, C and D (Figure 7) for each of the considered design variations. The vortices in the simulations of the staggered bed protection detach approximately every 600 seconds, which causes the cyclic behavior of the velocities. The exact cause of these vortices is unknown, but it is apparent that the geometry of the weir design greatly influences the local flow patterns.

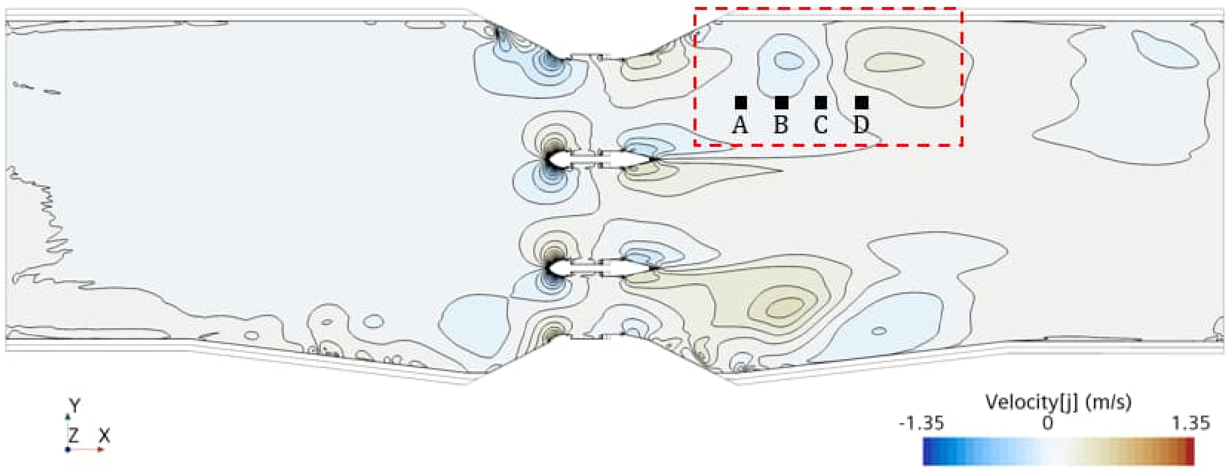

Figure 7. Instantaneous transverse velocities in a horizontal cross-section at NAP + 10 m for the 'Staggered' simulation. The region near the bank, just downstream of the weir, where vortices develop is marked with a red dotted rectangle.

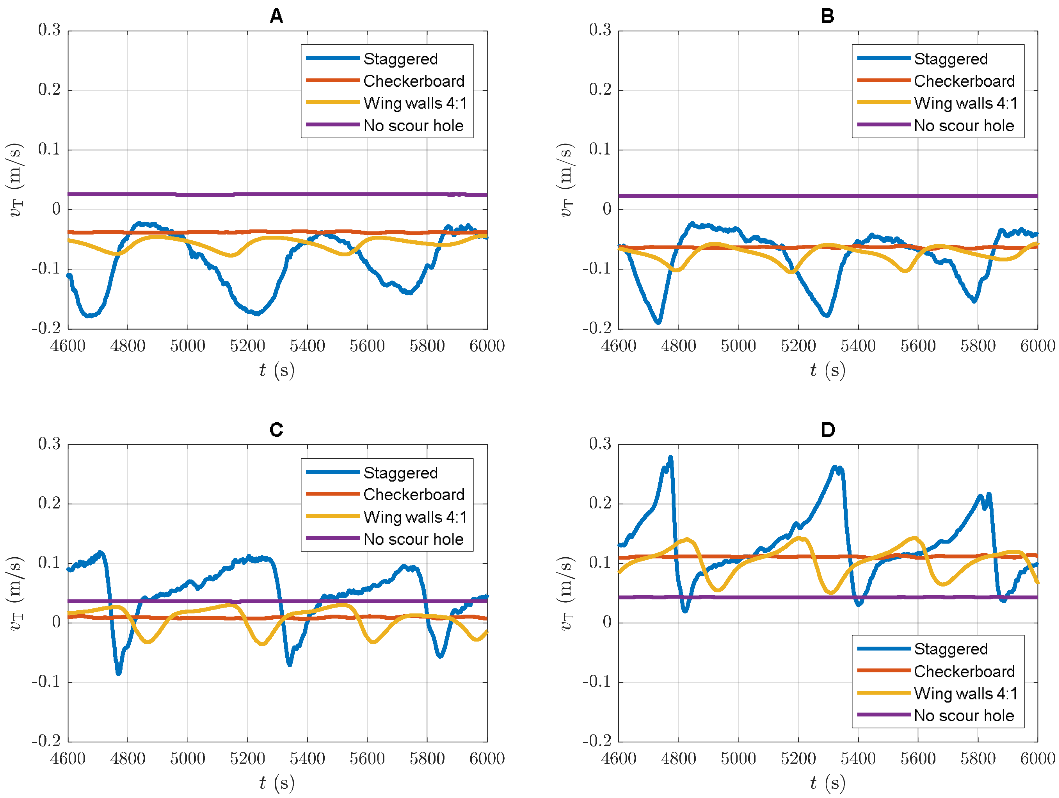

Figure 8. Maximum transverse velocities which occur along the draft of the design vessel at locations A, B, C, and D (in the middle of the weir opening for upstream going vessels, see Figure 7) for each of the considered geometry variations.

In case of the checkerboard bed protection schematization, a smaller horizontal recirculation zone is observed in combination with (nearly) an absence of perturbations. The vortex shedding appears to have been suppressed, resulting in a mostly stationary vortex. From the results presented in Figure 8, it is observed that adjusting the slope of the wing walls (4 to 1) also leads to (to a lesser degree) smaller velocity fluctuations than the original slope (2 to 1), and at a slightly higher frequency. Additionally, the vortices in the case of the adjusted wing walls appear to develop further downstream than in the original design due to the more gradual gradient in the lateral direction compared to the original design.

No instabilities arise in the simulation without a scour hole. Bed slopes parallel to the flow, like the scour hole present in all other simulation variations, amplify the growth of perturbations. Removal of the scour hole could therefore prevent these instabilities from developing. This phenomenon was studied extensively by Broekema et al. (2020). The velocity averaged over the conveyance cross-section reduces proportionally to the change of this cross-section which occurs at the location of the scour hole. As the mixing layer develops over the scour hole, two flow states could occur: 1) vertical flow attachment in combination with horizontal contraction (and thus, growth of the main recirculation zone) or 2) vertical flow separation in combination with horizontal divergence (and thus, a reduced horizontal recirculation zone) (Broekema et al. 2020). In the first case, the vertical stretching and horizontal compression of the conveyance cross-section could sustain high velocities in this part of the flow, whereas in the second case the quick divergence leads to rapid mixing and distribution of the flow. In the absence of a scour hole, the dynamics that could increase the effect of perturbations or the size of the recirculation zones are absent, thus leading to limited velocity fluctuations.

3.5. Discussion

The horizontal recirculation zones that develop downstream of the wing walls of the weir structure interact with the vertical distribution of the flow over the scour hole and significantly influence the quantities of interest for safe navigation through the weir. The perturbations which formed in the simulation with the checkerboard bed protection are smaller than those in the simulation with the staggered bed protection due to the increase in bed roughness and the resulting increase in turbulence and turbulent mixing in the vertical direction. The behavior of the flows given the different bed protection schematizations could be understood qualitatively and the checkerboard schematization appears preferable for nautical safety. However, no definitive conclusions can be drawn on how well the presented schematizations represent the bed protection downstream of weirs in the field, or whether the impact of the adjustment of the configuration of the roughness elements would be of similar magnitude in the field. (In a next step, field measurements carried out in 2023 at an existing weir will be used for validation.) Adjusting the geometry of the weir structure significantly influences the local flow patterns. The magnitude of the perturbations in the simulation with adjusted wing walls is smaller, the vortex shedding has a shorter period and vortices develop further downstream due to the more gradual gradient in the lateral direction compared to the original design. The absence of a scour hole prevents the formation of these instabilities, making this option the overall most preferable geometry variation from the perspective of nautical safety. In practice, however, removing the scour hole and maintaining the riverbed as such would require a significant extension of the bed protection structure.

Overall, it can be concluded that the horizontal recirculation zone downstream of the weir structure interacts with the vertical distribution of the flow over the scour hole and that these 3D effects have a significant impact on the quantities of interest for (safe) navigation through the weir.

4. Nautical Simulations

4.1. Objective

At an early stage, using the results of the CFD flow simulation with a smooth river bed as input, fast-time simulations have been carried out. These fast-time simulations aimed at gathering more insight on the nautical safety of a large vessel when passing through the weir opening. It showed that, given the conditions with a smooth bed and the corresponding strong eddies and transverse velocities near the left river bank, the passage of the vessel going upstream was not safe. Then, it was decided to carry out more extensive real-time simulations for multiple nautical scenarios, also including an adapted design of the weir, to determine whether the passage under design conditions satisfies the safety requirements.

4.2. Scenarios

The starting point of the simulations is a database with the following information:

- The surroundings of the weir: bed levels, bank contours, water levels, flow field from CFD, wind speed, Electronic Nautical Chart (ENC), Visual 3D-presentation of the weir and its surroundings.
- The available maneuver model of the vessel: push-tow unit (l x b x d: 191 x 11.4 x 3.5 m^3).
- Scenarios: vessel start position and start time, sailing lines, sailing speeds, vessel loaded or unloaded.

Presuming a river discharge of 1400 m^3/s, the flow field from the 'Staggered' flow simulation is used as input for the nautical simulations. In addition, the longitudinal force on the vessel which is due to the water slopes along the river and at the weir is described as a position-dependent force. Simultaneously, a characteristic wind force from the south-west (5 Bft, 5%-probability of exceedance) is applied; the wind speed that is experienced by the vessel is 8.25 m/s. The effect of the wind is maximum in scenarios with an unloaded vessel. To simulate the worst case, it is assumed that the vessel cannot use the bow thruster due to a technical failure. Initially, 24 nautical scenarios have been tested in the real-time simulator: two experienced captains, going upstream or downstream, with a loaded or an unloaded vessel, and starting at three different points in time to account for the nonstationary flow and the period of the vortex shedding at the bank. The captains should determine their sailing speed themselves. In consequence of the first results, multiple scenarios were added applying reduced wind conditions, other sailing speeds and bad visibility. The simulations have been carried out on the Full Mission Bridge of MARIN using DOLPHIN Version 2023.10, which is simulation software developed by MARIN using the eXtensible Modelling Framework (XMF) simulation technique. A scenario is considered unsafe when the captain uses, simultaneously, large rudder angles (20º) and a high rpm of the propeller (Half Ahead), for longer than one minute. At the same time, for a safe scenario, the distance between the side of the vessel and the nearest pillar or abutment must be larger than half the beam of the vessel (0.5b). The total width of the sailing path is an important feature characterizing the stability of the vessel.

4.3. Results

The main results of the real-time simulations carried out by both captains are the following:

- Going upstream, loaded vessel: the captains had no problem handling the hindrance caused by the horizontal recirculation downstream of the abutment. However, when the vessel was in the weir opening, the captains had to pay attention to the transverse flow coming from the upstream side of the abutment. This could be solved by further streamlining of the abutment.
- Going downstream, loaded vessel: as the flow is more symmetric, this does not present a problem. Although the vessel in the flow is manageable, it has to assume a drift angle to counteract the wind. Therefore, as the sailing path becomes wider, the vessel has to pass in the center of the opening.
- Going upstream, unloaded vessel: because of the reduced draft, the recirculation is less felt, and the wind force is dominant. The vessel is manageable, but the sailing path becomes too wide and the distance to the pillars too short. Passing through the weir, the captains temporarily reduce the drift angle, but past the weir opening the drift angle becomes large again, the sailing width increases to 40 m and the vessel may overlap the path of an oncoming vessel. Fortunately, the vessel has enough time to anticipate due to the low speed over ground.
- Going downstream, unloaded vessel: due to the cross wind and the high speed over ground the sailing path is very wide, up to 40 m. Right before the weir, the captains temporarily reduce the drift angle to pass the weir (Figure 9). The speed over ground is very high: 6.4 m/s. As a result, the vessel drifts with high speed towards the pillar on the right. Just past the weir opening the captain resumes the drift angle and the sailing path increases again to maximum 40 m. In all, it is concluded that it is a stressful and unsafe maneuver in an opening that is too narrow. An opening width between 46 and 50 m, based on additional calculations and expert judgment by the captains, respectively, is required to ensure a safe passage.

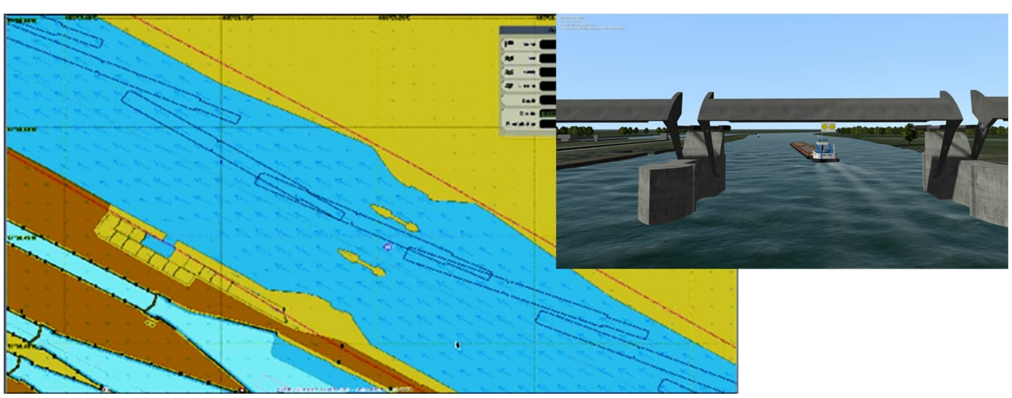

Figure 9. Real-Time simulation: going downstream with empty vessel, using the middle opening, during strong cross wind.

5. Conclusion

Currently, four of the Poirée-weirs in the Meuse, with an opening width between the pillars of circa 60 m, are navigable during river floods. However, when two vessels encounter at the open Poirée-weir, the vessel going upstream sometimes has to give right of way to the vessel going downstream. Due to a change of usage and river dimensions, the flow velocities in the Poirée-weirs have increased to 2.5 m/s. At the weirs in the Lower Rhine, the flow velocities are lower (ca. 1.5 m/s) and each sailing direction has its own opening of minimal 38 m wide at the maximum headway. That 38 m has been chosen as a starting point for the design of a new weir in the Meuse, which has two navigable openings of 38 m and one control opening of 24 m. It is clear that a safe passage of the weir depends on both the weather conditions, such as strong winds or conditions that cause poor visibility, and the hydraulic design of the weir and the bed roughness behind the weir sill. In this study, flow fields resulting from CFD simulations indicate that horizontal recirculation zones form downstream of the designed weir structure. These recirculation zones interact with the vertical distribution of the flow over the scour hole and influence the quantities of interest for safe navigation through the weir, particularly the transverse velocity gradients. Reducing the slope of the wing walls, which allowed for a more gradual transition in the lateral direction, and removal of the scour hole were shown to reduce the size of the vortices and improved the conditions for nautical safety. Subsequently, real-time nautical simulations based on the abovementioned flow fields showed that the openings are not wide enough to ensure a safe passage through the weir. The recirculation zones caused by the geometry of the weir did not cause too much hindrance. However, in the case of an unloaded vessel, especially going downstream with high speed over ground, strong winds resulted in too wide sailing paths. So far, it is concluded that for a safe passage an opening width between 46 m and 50 m is required.

6. ACKNOWLEDGEMENTS

This study has been carried out in a close collaboration between Rijkswaterstaat, Deltares and MARIN. The authors would like to express their appreciation to Rijkswaterstaat for their permission to publish the model results.

7. REFERENCES

Broekema, Y.B., Labeur R.J., and Uijttewaal, W.S.J. (2020). “Suppression of vertical flow separation over steep slopes in open channels by horizontal flow contraction. *Journal of Fluid Mechanics*, 885.

Carney, S.K., Bledsoe, B.P., and Gessler, D. (2006). “Representing the bed roughness of coarse-grained streams in computational fluid dynamics.” *Earth Surface Processes and Landforms*, 31, 736-749.

Gaay, A.C. de and Blokland, P. (1970). “The Canalization of the Lower Rhine.” Rijkswaterstaat Communications Nr 10, The Hague.

Hove, D. ten (2023). “Doorontwikkelen Richtlijnen Vaarwegen – doorvaarbaarheid stuwen in de Maas, real-time manoeuvreersimulaties.” MARIN, 34293-2-MO-rev.1, Wageningen.

Konijnenburg, E. van et al. (1912). “Rapport van de Nederlandsch-Belgische Commissie, ingesteld tot onderzoek van de kanalisatie van de gemeenschappelijke Maas.” Drukkerij Mouton&Co, The Hague.

Loor, A. and Kortlever W.C.D. (2018). “Determining Flow Velocities at Damaged Weir of Grave Using CFD.” Proc. 7th International Symposium on Hydraulic Structures, IAHR, Aachen.

Maijvis, S.D., Zubova, A., O`Mahoney, T.S.D., and Boschetti, T. (2023). “CFD Modelling of Flow Fields Through an Open Weir, Bed Protection Schematisation and Geometry Variations.” Deltares, 11209211-011-GEO-0002, Delft.

Schot J.W., Lintsen H.W., Rip A. and Albert de la Bruhèze A.A. (1998). “Techniek in Nederland in de twintigste eeuw. Deel 1. Techniek in ontwikkeling, waterstaat, kantoor en informatietechnologie.” Stichting Historie der Techniek, Walburg Pers, Zutphen.

Vries, J.W. de, Egelie C.F., and Jansen P.P. (1935). “Moderne beweegbare stuwtypen. Grootste afmetingen welke in elk type bereikt werden. Inrichting der vaste...” *Proc. XVIde Internationaal Scheepvaartcongres*, Internationale Permanente Vereeniging voor Scheepvaartcongressen, Brussels.